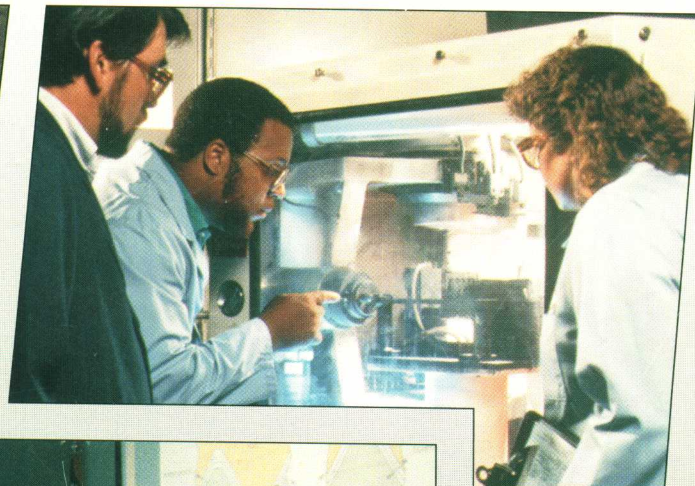
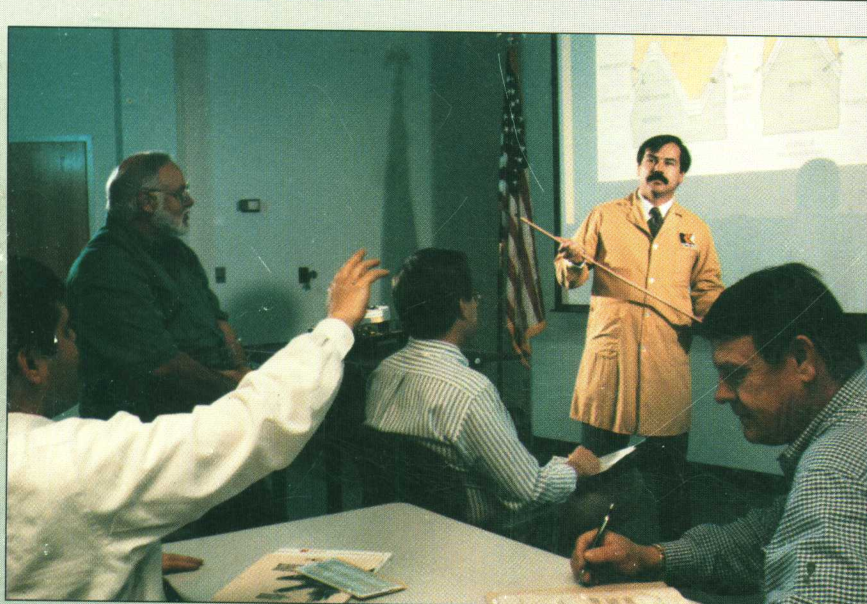
MACHINING TRAINING MANUAL





KENNAMETAL®

THE TABLE OF CONTENTS

TRAINING MANUAL

TURNING

INTRODUCTION

Turning accounts for nearly 25% of world tool consumption for metal cutting operations. Therefore, it is by far the most common machining method in use today. The process is simple, it involves the removing of metal from the outer diameter of a rotating workpiece in order to create a circular shape through the use of a single point tool. Under ideal operating conditions, the tool is used to shear the metal from the part in short distinct chips which are easily reclaimed, melted and reused.

In the early 1900's turning tools were solid rectangular pieces of HSS (high speed steel) with rake and clearance angles ground on one end, which were clamped directly in the tool post of a turning machine. When the tool became dull, the machinist often resharpened the tool on a pedestal grinder. Carbide cutting tools were introduced as small pieces or blanks which were brazed to a steel shank and ground with the same rake and clearance angles as those found on the original HSS turning tools. The increased hardness and wear resistance of these new tools provided improvements in productivity and tool life which benefited all of the major metal cutting industries. The greatest draw back to brazed carbide tooling was the expense in expertise and diamond wheels required to resharpen and maintain these tools. It became impractical for machinists to sharpen carbide tooling since the time required to grind an acceptable cutting edge was much greater when compared to HSS due to the significant difference in hardness. All these additional costs, however, were offset by the dramatic increase in productivity which resulted through the use of the carbide tools.

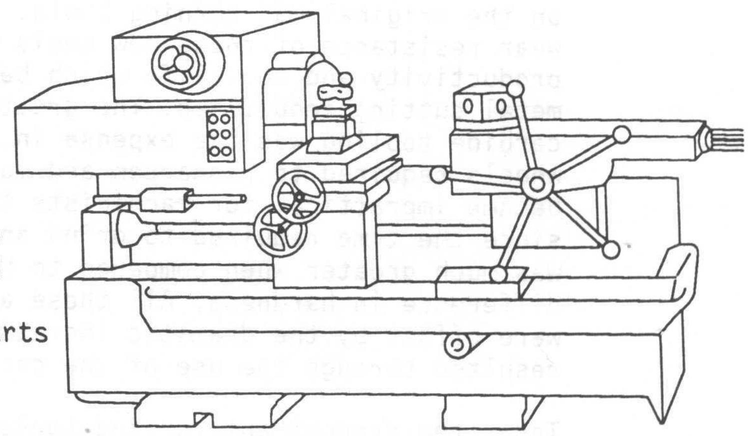
Today the predominant turning tools in use are coated carbide inserts which are mechanically locked in steel toolholders. These tools have almost entirely replaced the once common brazed carbide stick tools. The inserts are manufactured to precise tolerances with several available cutting edges. When a cutting edge becomes dull today, the machine operator simply unclamps and indexes the insert to the next available sharp cutting edge and reclamps the insert in the toolholder. Hence the name "indexable insert" or "indexable tooling". Our discussions on turning will be exclusively centered around indexable insert tooling.

TURNING MACHINES

The turning process involves the rotation of the workpiece and the linear travel of the tool along (turning operations) or across the workpiece (facing, grooving and cutoff operations). Turning machines are generally one of two styles, either horizontal or vertical, indicating the direction of the rotational axis of the workpiece. The turning machine employing a horizontal working axis is generally referred to as a lathe, while its vertical counterpart is commonly called a VTL (vertical turning lathe). These machines are frequently tooled with single point square shank indexable turning tools. Lathes can be either manually operated or CNC controlled. The following drawings show the differences between turning machines:

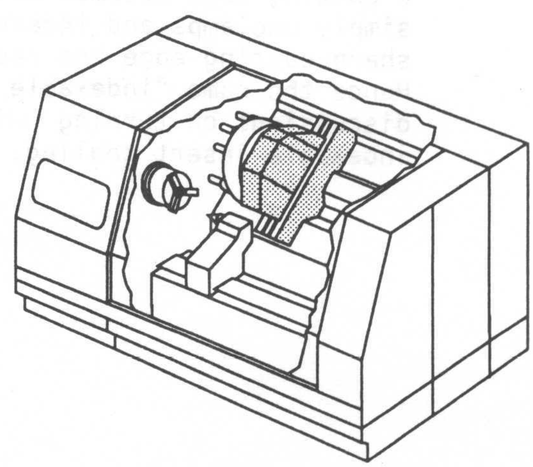
MANUAL LATHE

The manual lathe shown at the right is typical of lathes which are still used in tool rooms or in very low production applications. These machines are the gear driven predecessor of the NC lathes in use today. The major difference between these machines and NC lathes are accuracy, productivity and complexity. Manual lathes can machine many of the parts which are currently manufactured with NC equipment, but rarely with the same efficiency or speed.



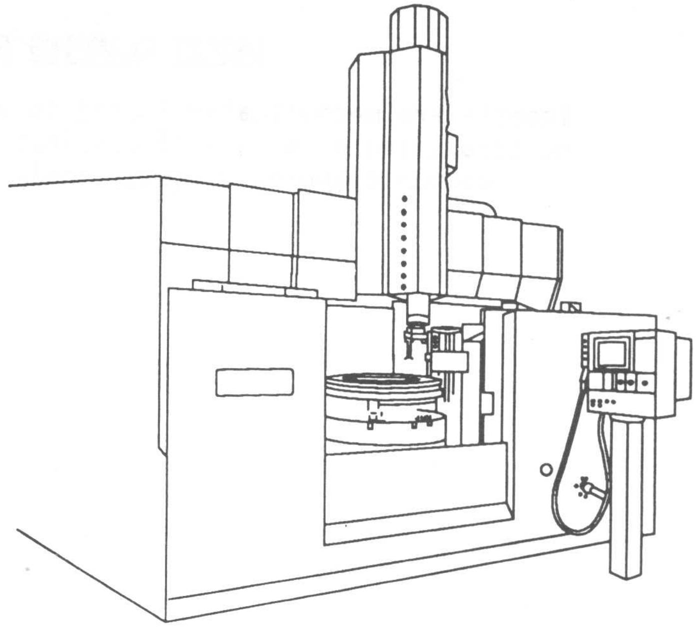
CNC LATHE (2 AXIS)

The greatest difference between a manual and CNC lathe is the number of tools which can be mounted in the machine, and the fact that the positioning of the tool point is done automatically by computer control. The machine turret houses eight to 16 tools in a single setup, depending on the machine used. CNC machines are normally totally enclosed with sheet metal to prevent chips from escaping, as they are machined from the workpiece.



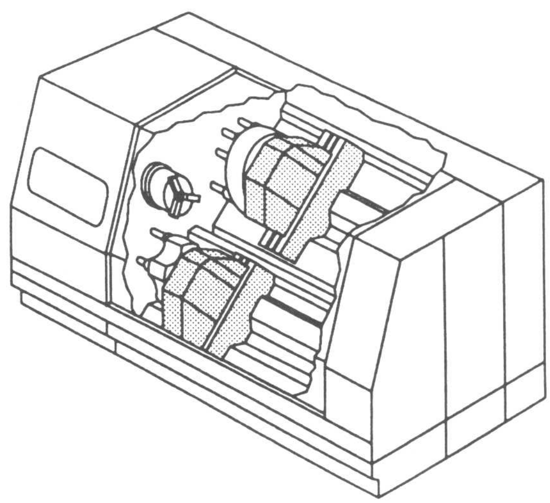
CNC VTL (2 AXIS)

The vertical turning lathe at the right is also CNC controlled with a turret full of tooling. The predominant difference between this machine and the CNC lathe is the chuck (1) orientation which is now vertical instead of horizontal. This is considered a 2-axis machine since the tool can move up and down (vertically) or across (horizontally) the workpiece.



CNC LATHE (4 AXIS)

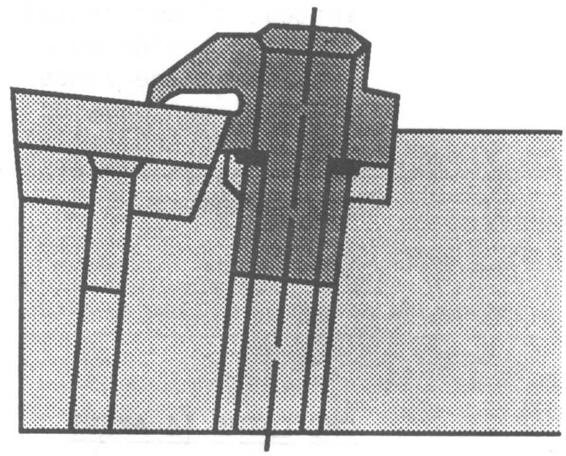
The CNC lathe detailed at the right is basically the same as the one mentioned previously except for one obvious difference. This machine has two independent turrets (1) instead of one. Since the upper moves independently of the lower turret, the machine is considered a 4-axis lathe, two axes per turret.



INSERT CLAMPING SYSTEMS

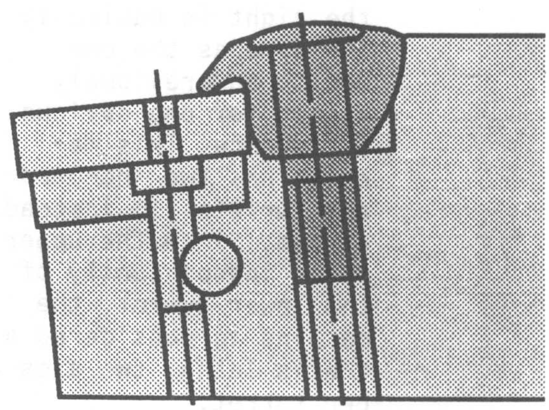
Inserts are mechanically locked in square shank tooling holders using a variety of distinct designs. Let's review the most common clamping systems, their merits and limitations:

C-CLAMPING



This method of clamping utilizes a top clamp on inserts without a hole. The system has limited application since it relies solely on friction to hold the insert in place. C-Clamping is normally used with positive rake inserts on medium to light duty turning operations.

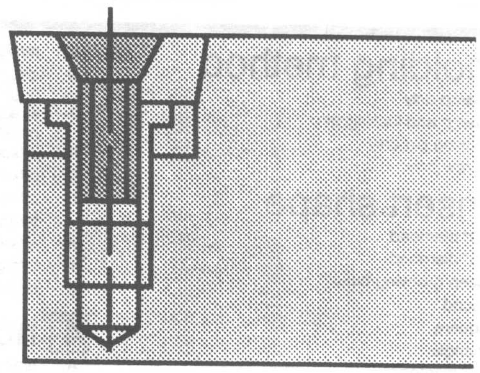
M-CLAMPING



Negative rake inserts with a hole are clamped using this design. The insert is forced against the pocket wall with a cam lockpin which secures the protective insert pocket shim. The top clamp is used to hold the back of the insert from lifting when a cutting load is applied on the insert nose. This design requires an excessive amount of hardware. This system is designed for medium to heavy turning operations.

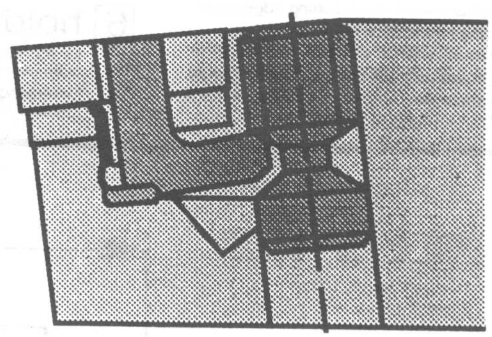
S-CLAMPING

This system merely takes advantage of a simple torx or allen head screw to secure the insert in the toolholder pocket. The inserts used in this system either have counterbores or countersinks. When indexing inserts in an application with intense heat the screws sometimes seize and are difficult to remove. The S-Clamping system is designed for light to medium turning operations.



P-CLAMPING

This style of clamping is the ISO std. for turning tools thus very popular in Europe and the Asian markets. The design principle is for negative rake inserts with holes. The insert is forced against the pocket walls using a toggle lever which tips as the lever screw is seated. This design is suited for medium to heavy turning operations but offers no means of preventing the back of the insert from lifting during the cut.



ANSI NUMBERING SYSTEM

The American National Standards Institutes system for numbering turning tool holders is as follows:

MTGNR-16-4-D

1 2 3 4 5 6 7 8

1 holding method

- C clamp only
- M clamp and locking pin
- P locking pin only
- S screw only

2 insert shape

- C 80° diamond
- D 55° diamond
- Q deep grooving/cut-off
- R round
- S square
- T triangle
- V 35° diamond

3 toolholder style

- A straight shank with 0° side cutting edge angle
- B straight shank with 15° side cutting edge angle
- C straight shank with 0° end cutting edge angle
- D straight shank with 45° side cutting edge angle
- E straight shank with 30° side cutting edge angle
- F offset shank with 0° end cutting edge angle
- G offset shank with 0° side cutting edge angle
- H threading and shallow I.D. grooving
- J offset shank with -3° side cutting edge angle
- K offset shank with 15° end cutting edge angle
- L offset shank with -5° side or end cutting edge angle
- M straight shank with 40° side cutting edge angle
- N straight shank with -27 1/2° end cutting edge angle
- P straight shank with 27 1/2° side cutting edge angle
- R offset shank with 15° side cutting edge angle
- S offset shank with 45° side cutting edge angle
- T offset shank with 30° side cutting edge angle
- V threading and shallow O.D. grooving
- W offset shank with 10° side cutting edge angle
- Y straight shank with 50° side cutting edge angle
- Z offset threading and O.D. grooving

4 holder rake

- A high positive
- N negative
- O neutral
- P positive

5 hand of tool

- L left
- N neutral
- R right

6 holder shank size

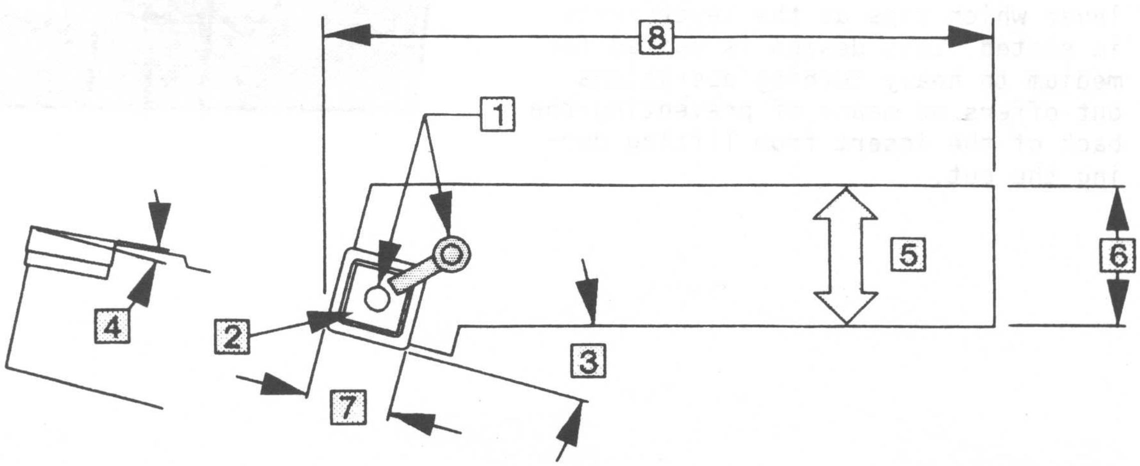
- for square shanks, this is the number of 1/16ths of an inch of width and height.
- for rectangular shanks, the first digit is the number of 1/8ths of an inch of width and the second digit is the number of 1/4ths of an inch of height—with the exception of 1 1/4" x 1 1/2" holder shank which has number (91).

7 insert I.C. size

- for inserts with 1/4" I.C. and over, this is the number of 1/8ths of an inch.
- for inserts with 1/4" I.C. and under, this is the number of 1/32nds of an inch.

8 shank qualification

- A qualified back and end, 4.0" long
- B qualified back and end, 4.5" long
- C qualified back and end, 5.0" long
- D qualified back and end, 6.0" long
- E qualified back and end, 7.0" long
- F qualified back and end, 8.0" long
- G qualified back and end, 5.5" long
- M qualified front and end, 4.0" long
- N qualified front and end, 4.5" long
- P qualified front and end, 5.0" long
- R qualified front and end, 6.0" long
- S qualified front and end, 7.0" long
- T qualified front and end, 8.0" long
- U qualified front and end, 5.5" long
- * all qualified surfaces are ± .003" over proper gage insert radius.

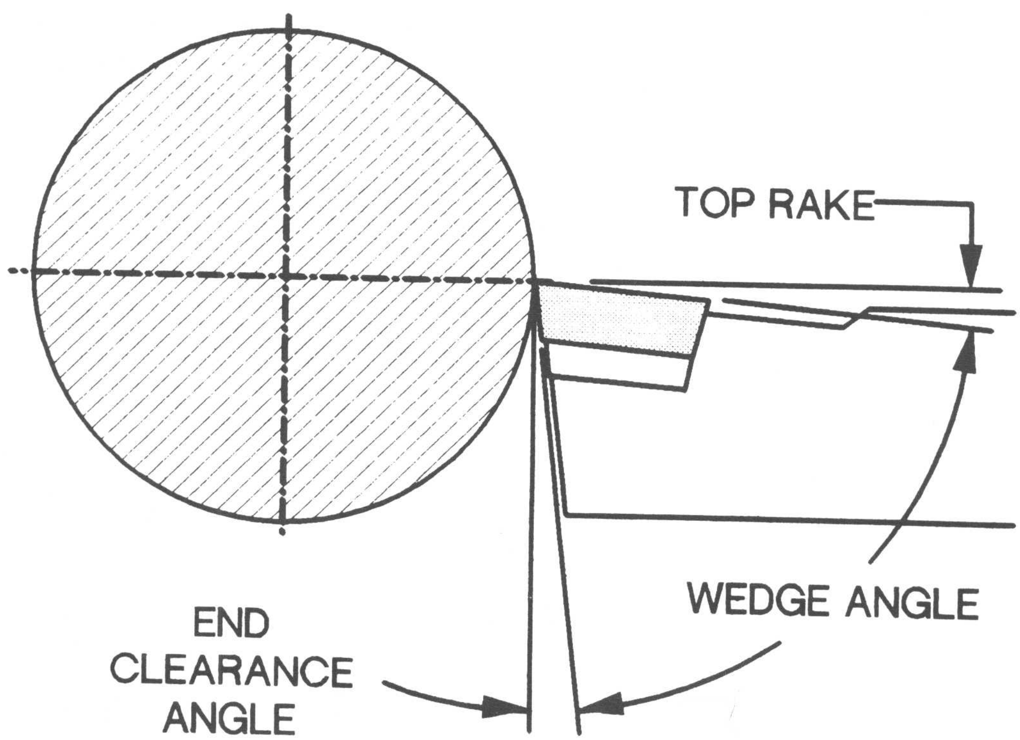


TURNING TOOL GEOMETRY

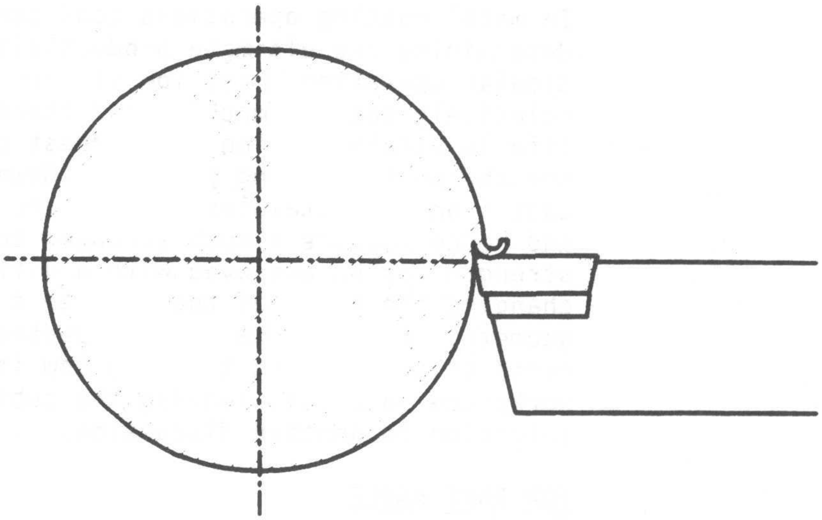
In metal cutting operations tool geometry plays a key role in determining the ultimate productivity and tool life of a particular operation. Most low silicon aircraft aluminum is relatively easy to machine and therefore the optimum tool life is attained using the hardest possible carbide grade and the sharpest shearing geometry. However, hardened steels, cast irons and stainless steels are more difficult to machine and hence require a much stronger tool. The additional strength can be achieved with a different carbide grade or by changing the shape of the tool to a stronger improved cutting geometry. For now, we will concentrate on the geometric aspects of the turning tool and how it applies to a specific workpiece material, leaving the subject of carbide grade selection to another discussion.

TOP RAKE ANGLE

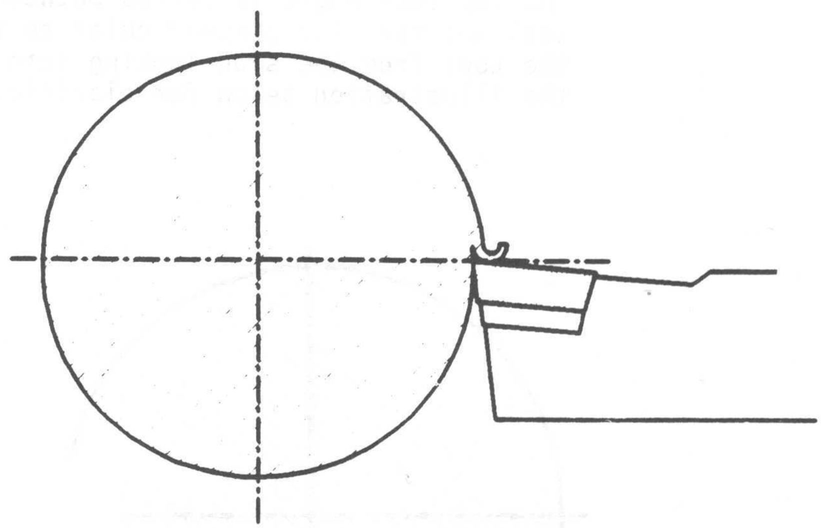
The top rake angle is formed between the face of the cutting tool and the line perpendicular to the workpiece when viewing the tool from the side looking into the end of the part. See the illustration below for clarification.



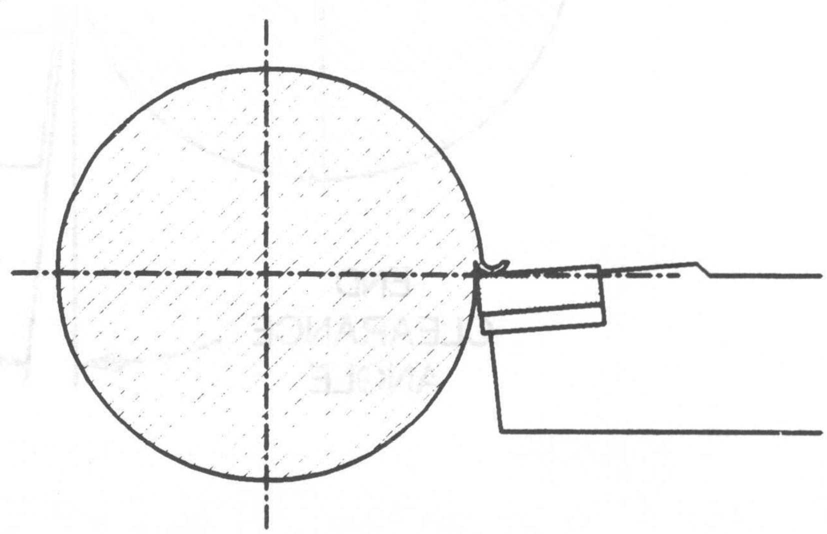
NEUTRAL TOP RAKE
The top rake is neutral when the sum of the end clearance angle and wedge angle is 90 degrees.



POSITIVE TOP RAKE
The top rake is positive when the sum of the end clearance angle and wedge angle is < 90 degrees.

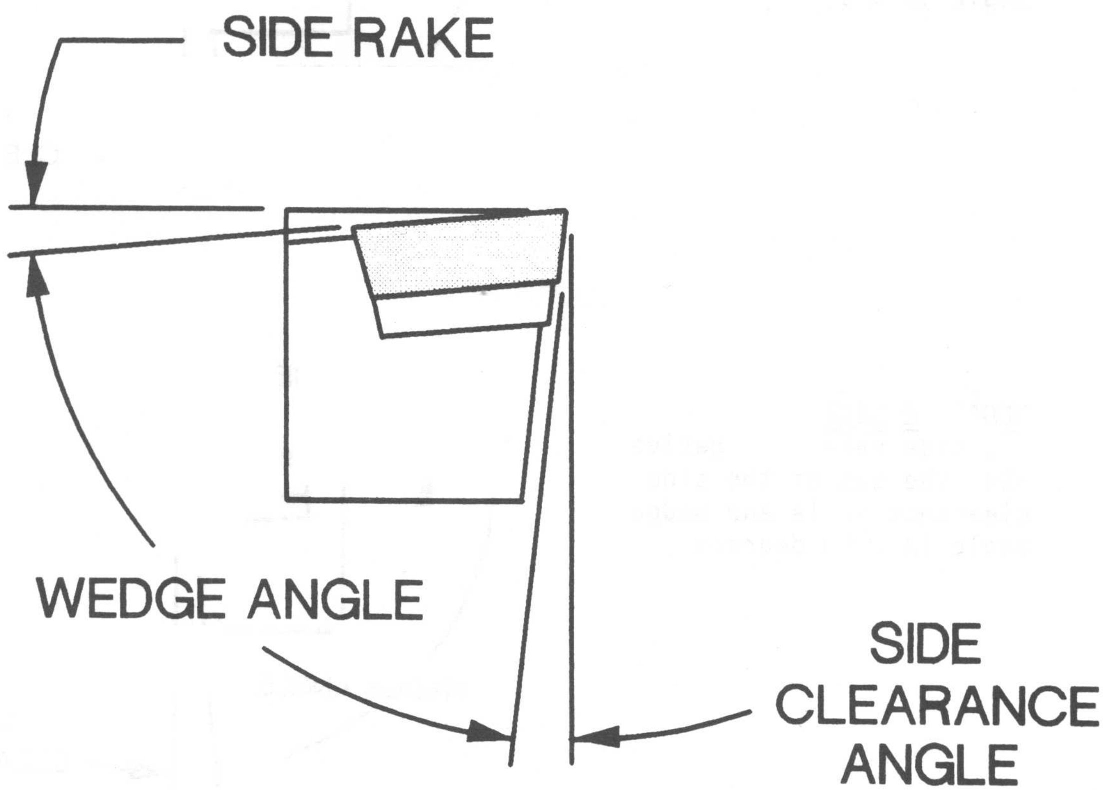


NEGATIVE TOP RAKE
The top rake is negative when the sum of the end clearance angle and wedge angle is > 90 degrees.

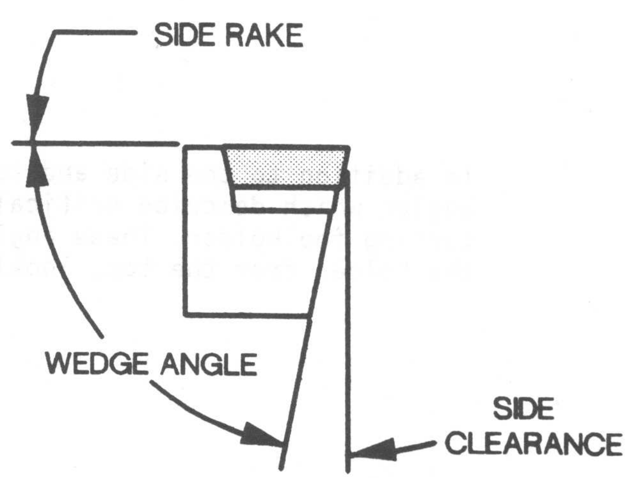


SIDE RAKE ANGLE

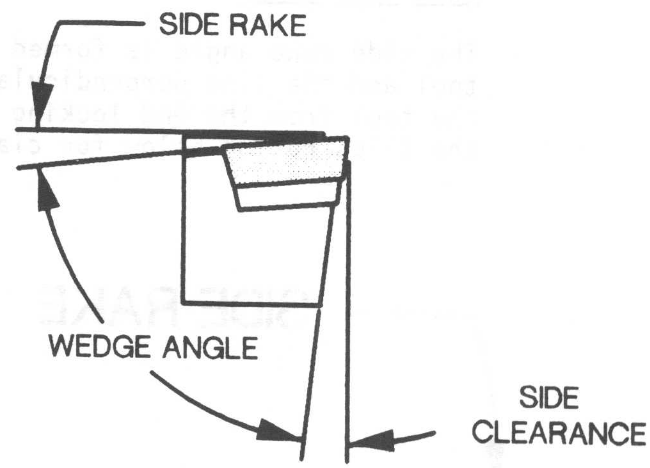
The side rake angle is formed between the face of the cutting tool and the line perpendicular to the workpiece when viewing the tool from the end looking at the side of the part. See the illustration below for clarification:



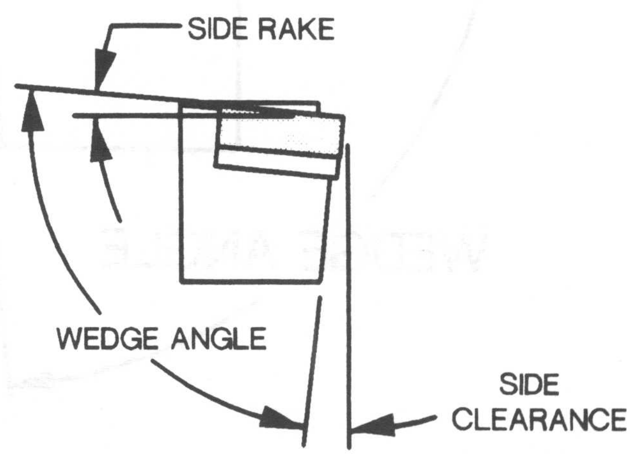
NEUTRAL SIDE RAKE
The side rake is neutral when adding the side clearance angle and wedge angle is 90 degrees.



POSITIVE SIDE RAKE
The side rake is positive when the sum of the side clearance angle and wedge angle is < 90 degrees.

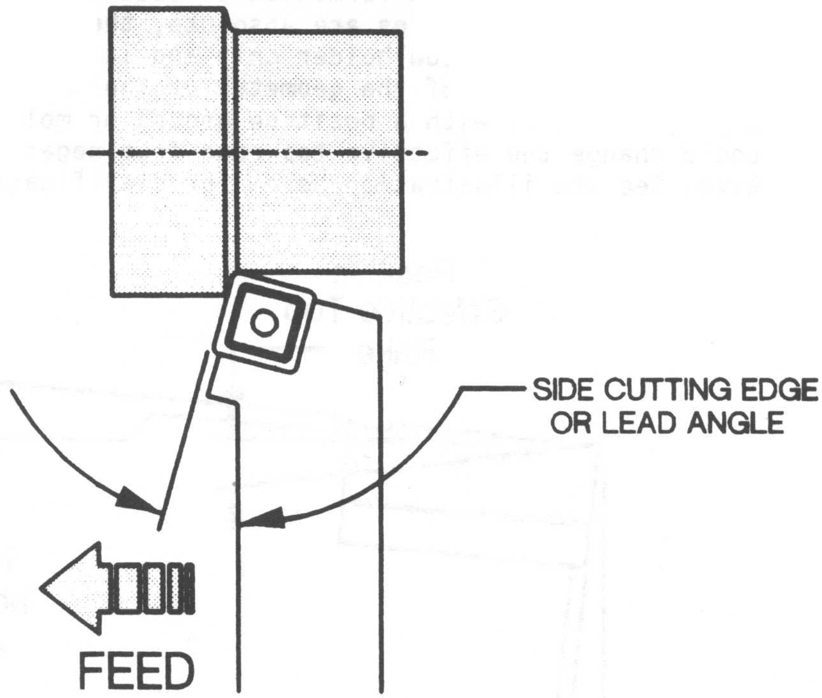


NEGATIVE SIDE RAKE
The side rake is negative when the sum of the side clearance angle and wedge angle is > 90 degrees.

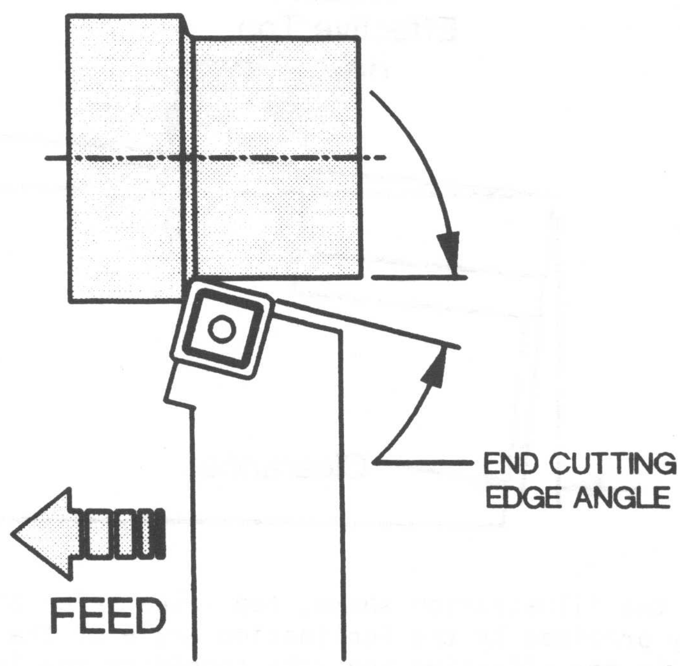


In addition to the side and top rakes there are two other angles which describe critical geometry on the ANSI style turning toolholder. These angles are evident when viewing the holder from the top, looking down on the insert face.

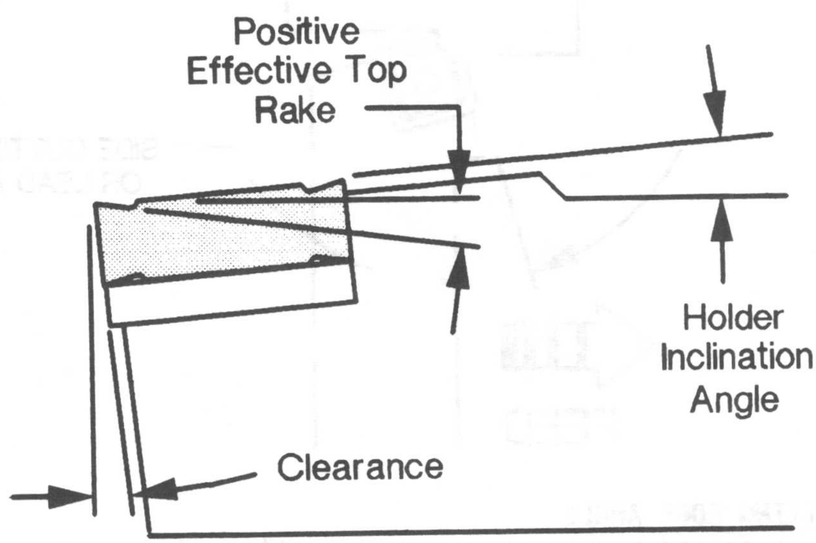
SIDE CUTTING EDGE OR LEAD ANGLE
The angle formed between the insert side cutting edge and the side of the tool shank. This angle leads the tool into the workpiece.

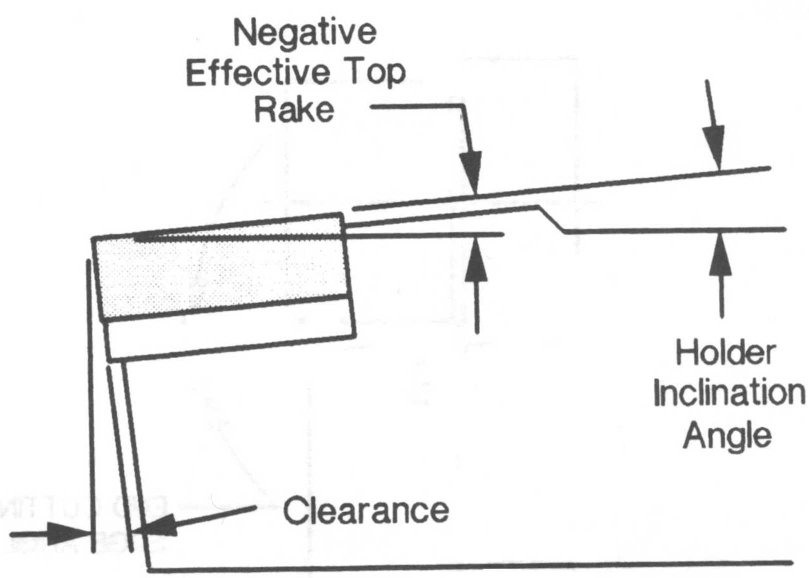


END CUTTING EDGE ANGLE
The angle formed between the insert cutting edge on the end of the tool and a line perpendicular to the back side of the tool shank.



It is important to note that the clearance angles shown on the end, as well as the side of the tools, are required to allow the tool to enter the cut. Without clearance, it would be impossible for chip formation to occur. While the end and side cutting edge angles are absolute, the top rake described earlier is for the toolholder only and it doesn't take into account the effect of the geometry of the insert. For example, an insert with a positive ground or molded chipbreaker could change the effective top rake from negative to positive. See the illustration below for clarification:





As the illustration shows, top rakes refer only to the geometry provided by the inclination angle of the toolholder, while the effective top rake considers the insert chip groove geometry combined with the toolholder inclination angle.

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 7 basic tool styles outlined by the ANSI system. The designations for the 7 primary tool styles are A,B,C,D,E,F and G. Of the 7 styles identified, the following 3 are not widely used:

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.

The more commonly used toolholder styles of the 7 mentioned are as follows:

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

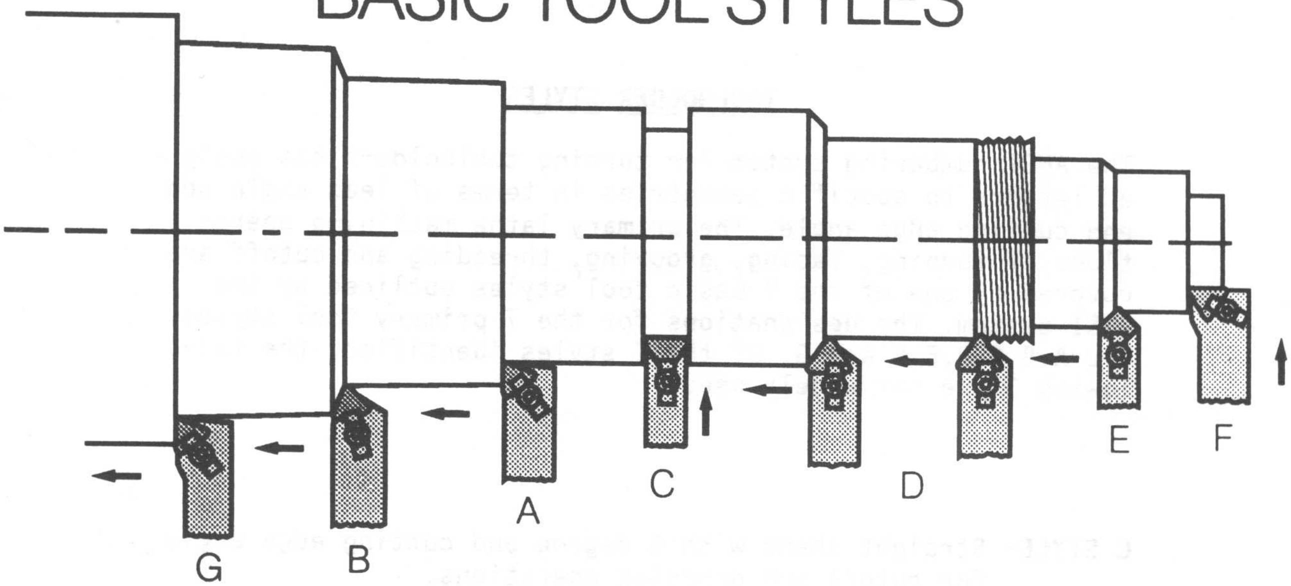
B STYLE- Straight shank with 15 degree side cutting edge angle, for turning operations.

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.

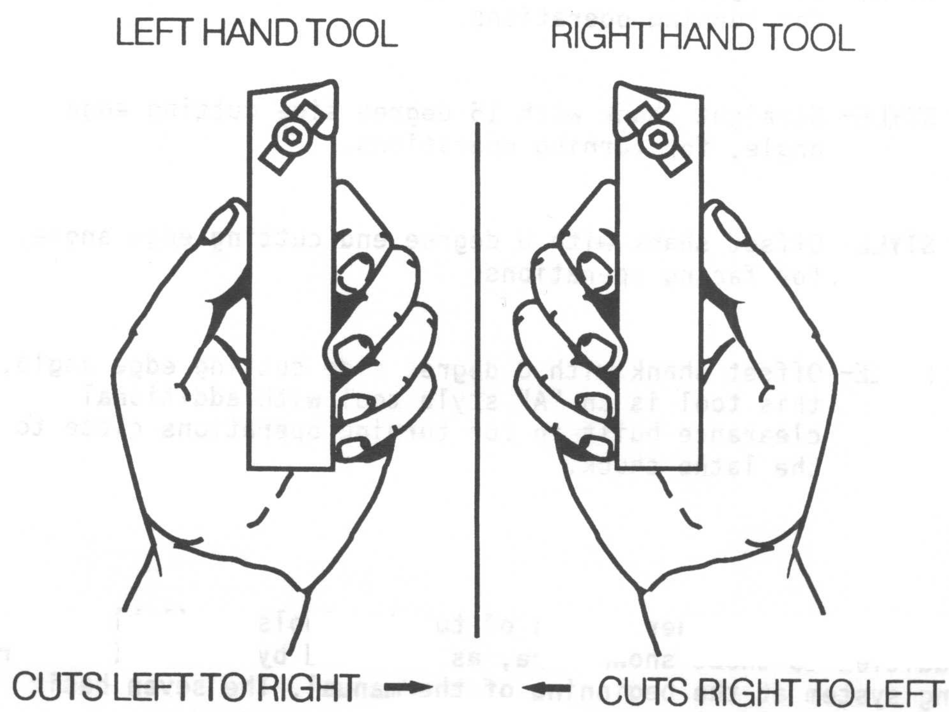
There are many other styles of turning tools available in addition to those shown here, as detailed by the ANSI numbering system at the beginning of the manual. The seven basic tools are shown in operation on the next page.

BASIC TOOL STYLES



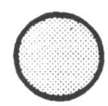
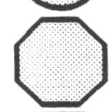
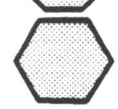
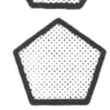
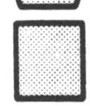
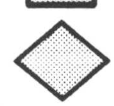
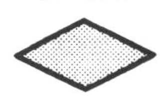
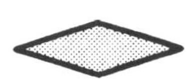
RIGHT AND LEFT HAND TOOLHOLDERS

The toolholder styles discussed on the previous page and shown above represent a fraction of those standard styles available from most indexable cutting tool manufacturers. ANSI std. 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 the picture below (insert facing upward), it will cut from right to left. The left hand tool, when held by the shank as shown in the picture below (insert facing upward), will cut from left to right.



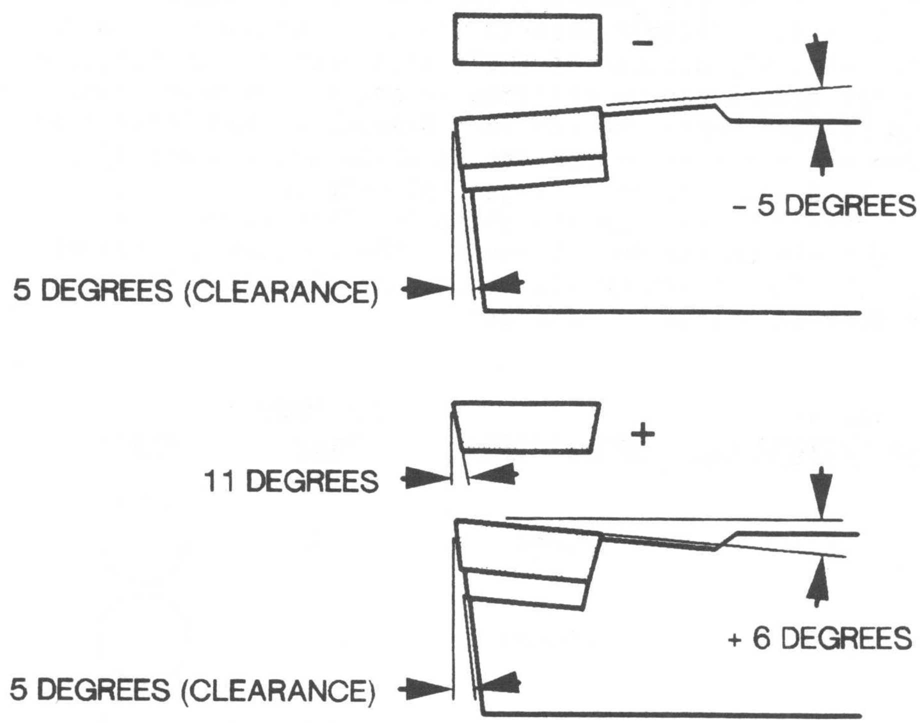
TURNING INSERTS

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 chip breakers. 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 inserts strength based on its shape is, "the larger the included angle on the insert corner, the greater the insert strength." The following list shows the different insert shapes from strongest to weakest:

INSERT LETTER DESIGNATION	DESCRIPTION	INCLUDED ANGLE	DRAWING
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	

POSITIVE AND NEGATIVE GEOMETRY

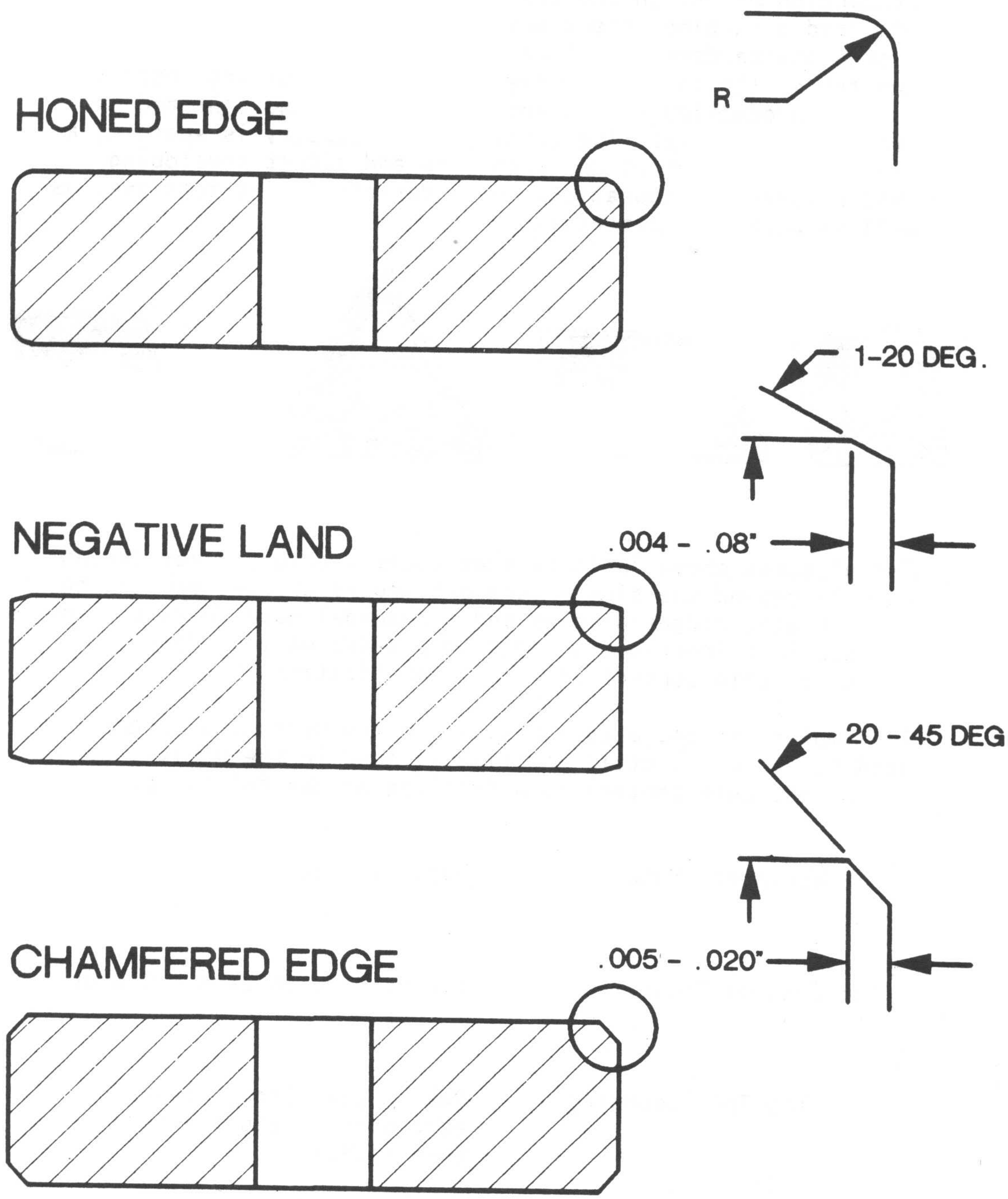
Beyond their individual geometric shapes inserts are categorized as being either positive or negative. Negative inserts have square sides (90 degree included angle) relative to the top face of the insert and, therefore, to obtain the appropriate cutting clearance, they must be mounted in toolholders with negative top and side rake angles. Positive inserts have angled sides. Thus, they can be mounted in toolholders with positive top and side rake angles. See the illustrations below:



EDGE PREPARATIONS

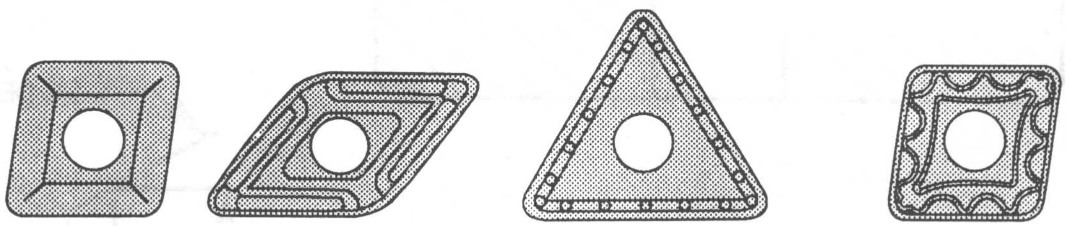
The performance of an indexable turning insert can be enhanced by altering the cutting edge during the final stages of the manufacturing process, from a perfectly sharp to a more blunt or rounded edge. These small alterations are often impossible to detect to the untrained eye, although in many cases, they can be the difference between success and failure in machining a difficult workpiece material. The bottom line is that edge preparations provide additional strength. The cutting edge can be prepared by honing, chamfering or applying a negative land.

Hones are nothing more then the rounding of all the insert cutting edges using mechanical methods. They are applied using rotating wire brushes, rotating hard rubber wheels or an abrasive mixture. Almost all of the inserts receiving coatings are honed first. Chamfers are ground on all the insert cutting edges from an angle of 20 to 45 degrees at a width of .005" to .020". Negative lands are ground on inserts from a few degrees up to 20 degrees. The width of a negative land can range from .004" to .08". Heavy negative lands are used under extreme machining conditions where interruptions and shock loading is prevalent. The land acts to change the direction of the cutting force from across the sharp edge through a larger cross-section of the insert. Heavy negative lands must be used under the appropriate cutting conditions, misapplication can lead to excessive cutting force and increased power consumption or vibration which will ultimately reduce tool life.



CHIP BREAKING THEORY

The extensive use of CNC turning machinery has increased the need for reliable chip control to maintain efficient production. In those cases where highly ductile workpieces are machined and poor chip control exists, long continuous strands of metal are created which wrap around tools and the part. This leads to premature failure due to chipping of the tool an excessive number of times to change parts, ultimately reducing the capacity of the machine tool by wasting precious production hours. In the past, many turning tool setups utilized a carbide insert mounted in an indexable holder with a separate hardened steel or carbide chip breaker clamped to the top of the insert. Today, most of the turning inserts used in production situations have pressed and sintered or ground chip grooves. The cutting tool industry is spending a significant amount of research time and effort developing chip breaker configurations for a variety of applications, as well as workpiece materials.

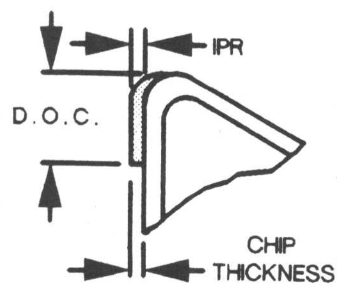


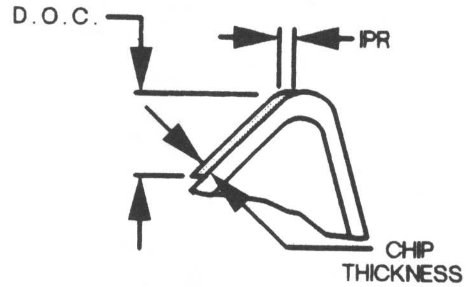
The pictures above indicate that today's chip breaker designs go well beyond the simple straight ground groove used in the past. Flats, ridges dimples and sinusoidal wave forms are now pressed into indexable inserts to provide many different options for chip control on turning applications.

In simplest terms, a chip breaker is the mechanical means used to force a ductile material to bend to the breaking point. Chip control is a function of the following:

Workpiece Material	Machinability
The Cutting Conditions	Speed, Chip thickness & width
The Tool Geometry	Rake angle, front clearance edge prep., chip breaker, nose radius

The strength of a ductile chip can be reduced significantly by increasing the cutting speed of the operation to aid in softening the material. In addition, the use of large lead angle tools will thin the chip, thereby reducing its strength. See the pictures below:



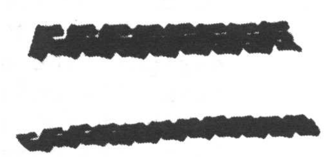


The key to chip breaking is to allow the chip to flow as freely and unrestrained as possible while still breaking it and thus, maintaining control. The force required to bend and break the chip consumes machine horsepower; therefore, the goal is to utilize the minimum required power to attain acceptable chip control. The following pictures illustrate unacceptable chip control:

IRREGULAR OR RIBBON SHAPED CHIPS
occur when an insert has too low a feed rate for a specific depth of cut, or too light a depth of cut for a particular chip breaker design. Since the chip has relatively low strength, it will not curl and break predictably.

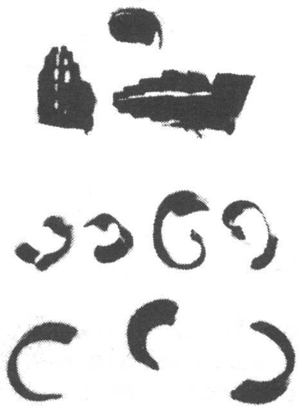
FLAT SPIRAL CHIPS
these chips are often a result of cutting an extremely light depth of cut, generally very close to the insert nose radius. To correct this condition: use a smaller nose radius, increase the depth of cut, increase feed rate or use a different chip breaker.

HELICAL AND LONG SPIRAL CHIPS
are a result of using a chip groove that is too wide for a specific feed rate and depth of cut. To correct this condition, either the feed should be increased or the chip breaker should be replaced with a narrower grooved style for lighter feed rates.



CRUMBLING CHIPS
this type of chip formation is almost always a result of over feeding the chip breaker.

Remember, the goal is to break the chip to a manageable size without creating any excess cutting force which could be detrimental to tool life. The following groups of pictures illustrate acceptable chip control:

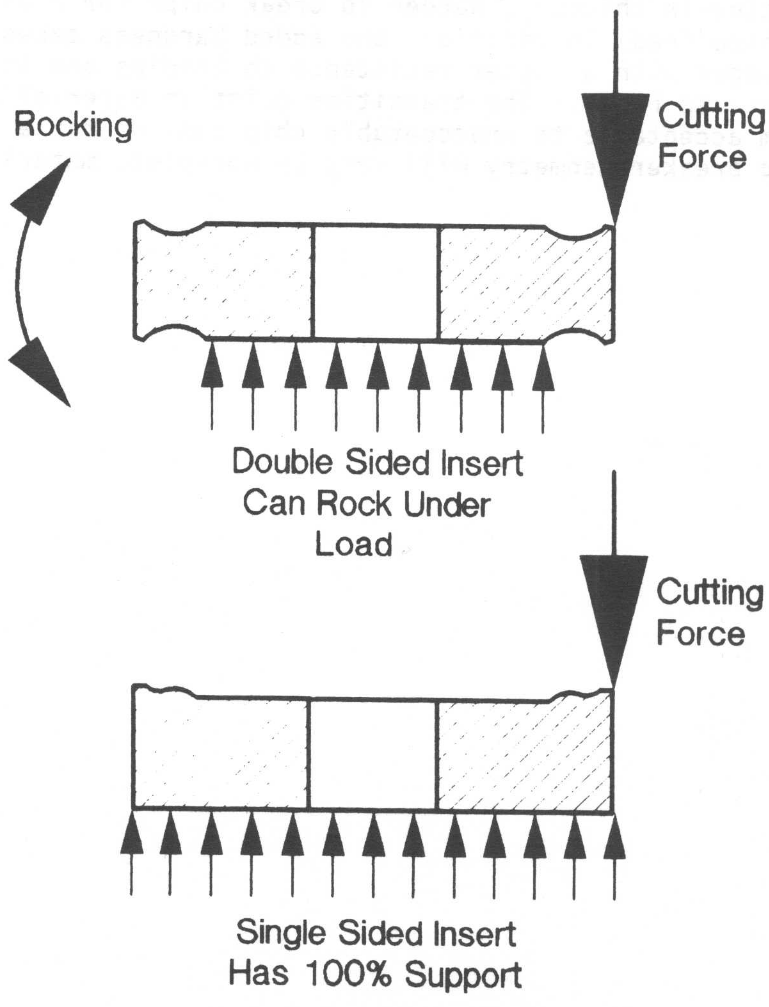


The forces created during chip formation are directly affected by how freely a chip is allowed to form. For example, the crumbled chips shown in the picture on the previous page result when the chip is crowded by the chip breaker. This causes the feed and radial cutting forces to rise significantly. When this situation is allowed to continue, the wall at the back of the chipbreaker will either erode, break away or cause catastrophic tool failure. Studies have shown that the shape of insert chip grooves have a larger effect on the radial and feed forces than on the tangential force component of a turning operation. This is why ground or molded insert grooves are often preferred over flat top inserts with mechanical chip breakers, since, for a specific set of cutting parameters and toolholder geometry, a ground or molded chip groove creates less vibration and tool force on the workpiece. In addition, on a continuously chipping material such as 1045 steel, tool life is often greater on a molded chip groove or ground insert when compared to a flat top insert with a mechanical chip breaker. The flat topped insert will usually crater at a greater rate, leading to deformation of the insert for a particular tool geometry, causing the chips to flow less freely over the flat top design. The depth of cut can also influence the effectiveness of a chip breaker's ability to control chips. When the depth of cut is approximately two times the insert radius or less, the chip breaker will exhibit different performance characteristics than if the depth were greater. In this type of situation, the feed required for adequate chip control will be greater than it would be at a larger depth of cut, since at least half of the chip is being thinned by the insert radius.

Chip control is also a function of the machinability of the workpiece material. Steels with carbon contents of .20% and less are highly ductile in an annealed condition (less than 18 Rockwell "C") and therefore they present chip control headaches especially when o.d. turning shallow depths of cut. When these material's are machined in a heat treated condition of 30-35 Rockwell "C", chip control becomes much more predictable. A materials increased hardness will lower its machinability rating in terms of tool life, but in reality, the added hardness is a benefit in the practical sense of chip handling and control. There is, however, a point where an added material hardness is detrimental to tool life and chip control even for low carbon steels. When this point is reached, the chip will not increase in thickness as it travels over the top rake surface of the tool to the same degree as it would at a lower material hardness level, resulting in thinner , harder to break chips for a given machine feed. In addition, the added hardness makes the chip stronger with a higher resistance to bending and is therefore harder to break. The transition point in material hardness from acceptable to unacceptable chip control for a specific chip breaker geometry will vary by workpiece material.

ONE- VS. TWO-SIDED INSERTS

Why would you ever select an insert with a chip breaker on just one side when there are so many inserts available with chip breakers on two sides? The key is in the type of turning operation you're trying to process. You'll notice that most of the single sided chip breaker geometries are recommended for roughing applications. Therefore, these inserts will be subjected to heavy tangential cutting loads and to prevent insert rocking and movement the non cutting side of the insert should be flat. The picture below illustrates that due to the chip breaker size and shape, 30-40% of the insert cutting edge is unsupported when a two sided chip breaker insert is selected instead of a single sided design.



Under severe cutting conditions, the use of two-sided inserts can lead to vibration, tool breakage or a compromise in productivity.

CALCULATING CUTTING SPEEDS AND FEEDS

The turning operation is a combination of linear (tool) and rotational (workpiece) machine movements. The rate in IPR (inches/revolution) that the tool travels along or across the workpiece is referred to as the machine feed. The SFPM (surface feet/minute) or speed at which the part surface rotates is known as the cutting or surface speed. These two important criteria are selected to either maximize tool life and productivity or to balance them.

The cutting speed in turning, is defined as the distance in feet traveled by a point on the part surface being machined in one minute. The formula normally used to calculate cutting speed is as follows:

SFPM = (Workpiece Circumference) x (RPM)

Where :

SFPM = surface feet per minute, or the distance traveled by a point on the workpiece periphery (being machined) in feet each minute.

Workpiece Circumference = the distance around the workpiece periphery (being machined) in ft.

RPM = revolutions per minute

In the case of a workpiece, the circumference is:

Workpiece Circumference = π/12 x (d) = .262 x d

Where :

Workpiece Circumference = the distance around the workpiece periphery (being cut) in feet.

π = is a constant, of 3.1416

d = the cutter diameter in inches

By substituting for the workpiece circumference, the cutting speed can now be written as:

SFPM = .262 x d x RPM

This formula can be used to determine the cutting speed at the periphery of any rotating workpiece on the surface being machined. For a more in depth explanation of cutting speed refer to the " MILLING MANUAL".

EXAMPLE #1
If you were finish turning a steel axle shaft to a 3.25" diameter at 1057 RPM, what cutting speed would this represent?

$$SFPM = .262 \times d \times RPM = .262 \times 3.25 \times 1057$$

ANSWER : SFPM = 900

EXAMPLE #2
What RPM would you select to maintain a cutting speed of 2000 SFPM on a cast iron brake drum with a finished outer diameter of 7.2"?

$$RPM = \frac{SFPM}{.262 \times d} = \frac{2000}{.262 \times 7.2} = 1060$$

ANSWER : RPM = 1060

The prevalent use of CNC lathes with constant surface speed control has virtually eliminated the need to manually calculate surface speeds. The machine operator or programmer can select a surface speed, and the machine will automatically determine and alter the RPM as the turning tool traverses the different diameters along the workpiece's outer profile. The ability to calculate cutting speeds, however, is a continued requirement to determine cycle times for part processes.

Once the cutting speed is selected for a particular workpiece material and condition, the appropriate feed rate must be determined. When we establish feed rates for turning tools, the goal in roughing applications is to attain the maximum metal removal rate possible with the available part rigidity and machine horsepower. In finish turning operations, feed rates are established to produce the surface finish specified on the part blueprint. Feed in turning is measured in inches per revolution, or IPR. This represents the linear distance the tool moves in inches for each revolution of the part. The feed is also expressed as the distance traveled in a single minute, or IPM (inches per minute). To calculate the feed in IPM, use the following formula:

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

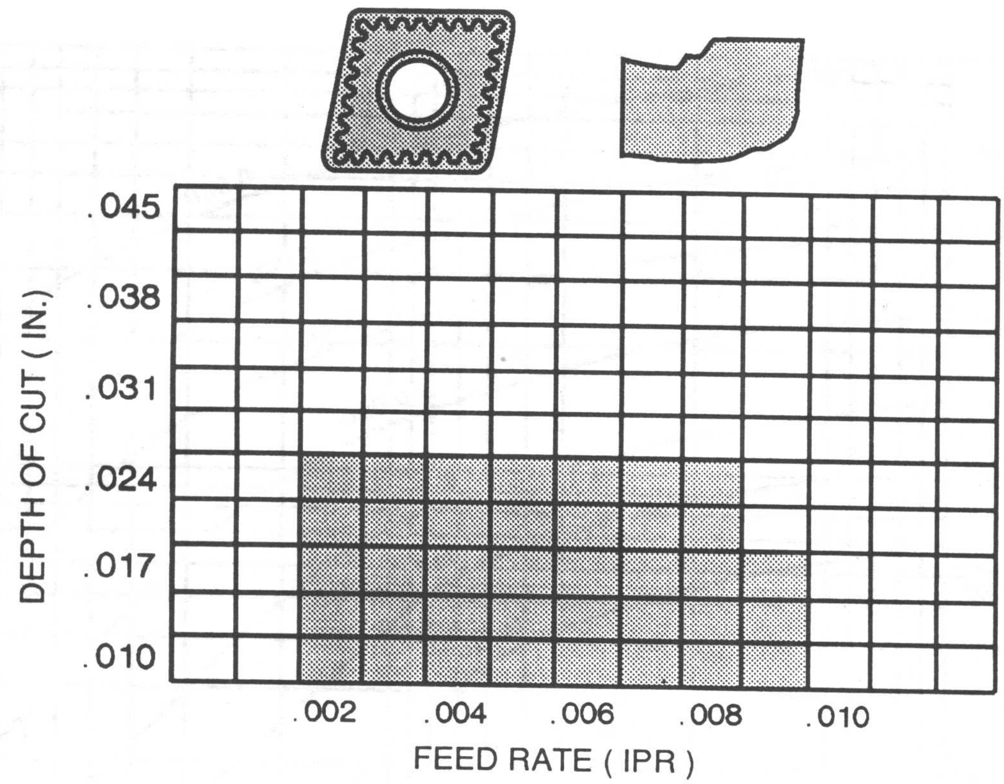
Where :

IPM = inches per minute

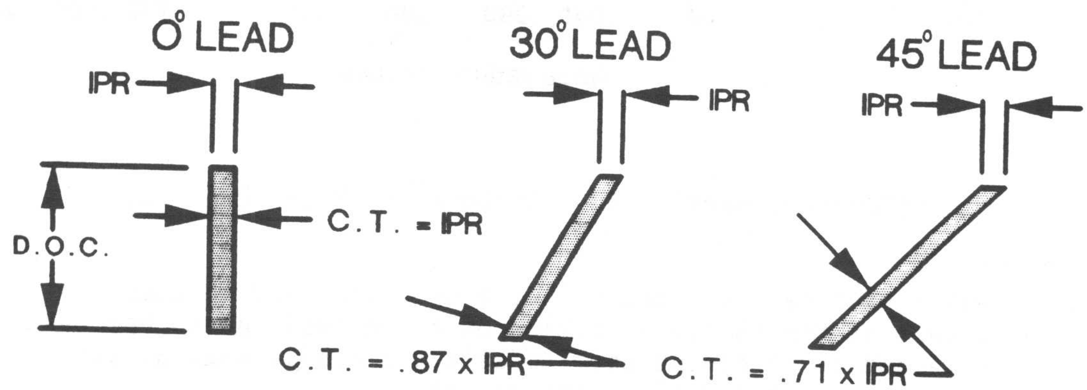
IPR = inches per revolution

RPM = revolutions per minute

The selection of feed rates is often based on prior test work and the establishment of a chip form diagram which indicates where acceptable chip control can be attained when using a specific chip breaker at various depths of cuts and IPR settings on a workpiece material. The picture below shows an example of a chip form diagram, with the shaded area highlighting where acceptable chip control exists:

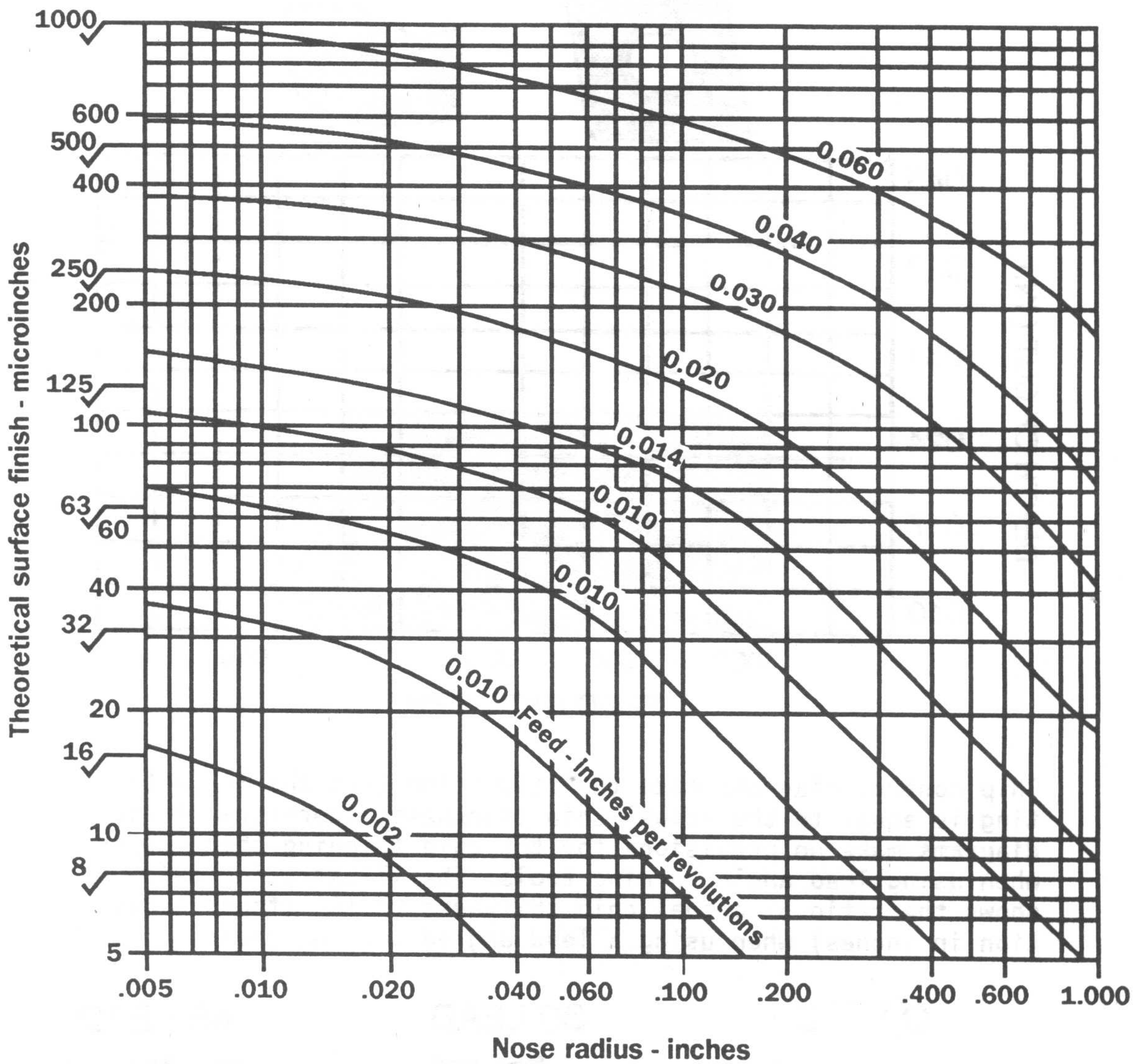


Chip control diagrams make the assumption that the IPR setting is equal to the actual chip thickness; therefore, these diagrams make no provisions for the chip thinning that occurs when using lead angle turning tools. The illustration below shows the ratio of actual chip thickness to IPR (feed/revolution in inches) when using a lead angled turning tool:



SURFACE FINISHES IN TURNING

The surface finish produced on the part during a turning operation is dependent on the rigidity of the tool, machine and workpiece, as well as the relationship between the machine feed (IPR) and the nose radius on the insert. To obtain the theoretical surface finish for a turning operation in microinches, use the following chart:



(Source : Mark's Std. Handbook For Mech. Engr.'s)

This chart can help you select a specific feed to attain a blueprint finish specification once the tool radius has been selected or it can be used to select a radius size given a machine feed and finish requirement.

CUTTING ANGLES & FORCE AND WORKPIECE MATERIALS

In general, the softer and more ductile the workpiece material, the greater the top rake of the turning tool. Ductile workpiece materials demand high positive shearing angles while harder tougher materials require more neutral or negative geometries. The following general recommendations are for the top rake angles normally associated with the materials listed:

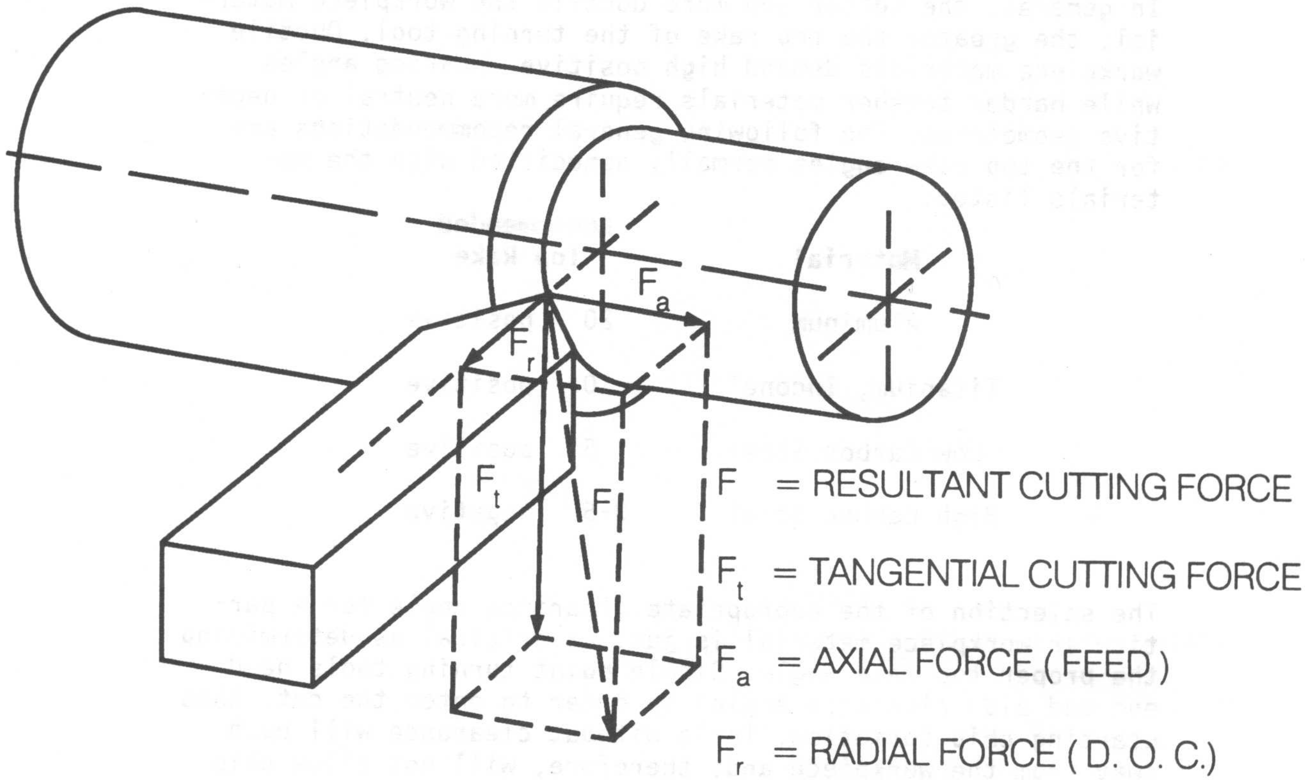
Material	Recommended Top Rake	
Aluminum	20	positive
Titanium, Inconel	10	positive
Low Carbon Steel	5	positive
High Carbon Steel	0-5	negative

The selection of the appropriate clearance angle for a particular workpiece material is just as critical as determining the proper top rake angle. Single point turning tools need end and side clearance angles in order to enter the cut, thus starting chip formation. Tools without clearance will push away from the workpiece and, therefore, will not allow chip formation to take place. The following list of general recommendations are for the clearance angles normally associated with the materials listed:

Material	Recommended Clearance
Aluminum	10
Titanium, Inconel	6
Low Carbon Steel	5
Alloy Steels	4

CUTTING FORCES IN TURNING

The tangential, radial and axial cutting forces are the three primary loads which act on the workpiece in a turning operation. The tangential force has the greatest effect on the power consumption of a turning operation, while the axial or feed force exerts pressure through the part in a longitudinal direction. Finally, the radial force tends to push the workpiece and toolholder apart. When these component forces are added together a resultant cutting force is established. The following picture illustrates the relationship of these forces as they act on a workpiece:

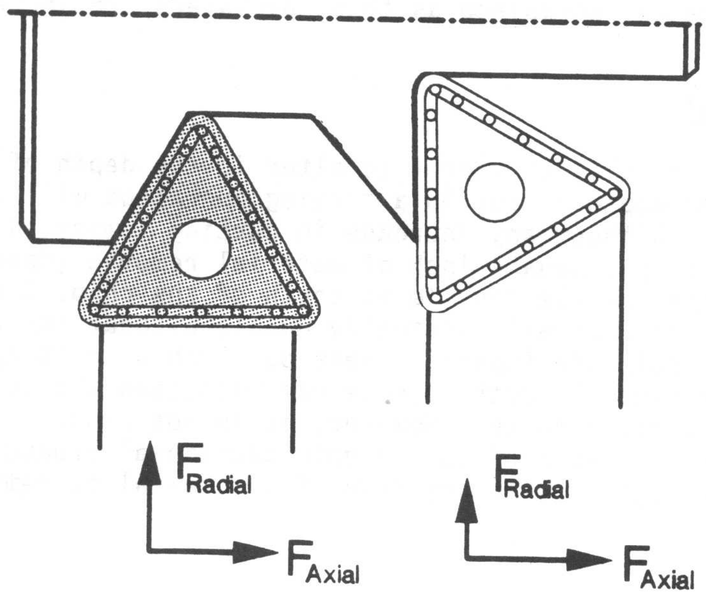


The ratio of the cutting forces in turning is, 4:2:1 for a zero degree lead angled tool. In other words, if the tangential force was 1000 lbs. then the axial (feed) force would be 500 lbs. and the radial force 250 lbs. This is a rough guideline, but it illustrates that the tangential force is always the largest component of the resultant cutting force. The axial force is normally the second largest of the three component forces and the radial is often the smallest component. The magnitude of the radial and axial force is altered as the toolholder lead angle is changed. The greater the lead angle, the larger the radial cutting force and the smaller the axial (feed) force in a turning operation.

LEAD ANGLE

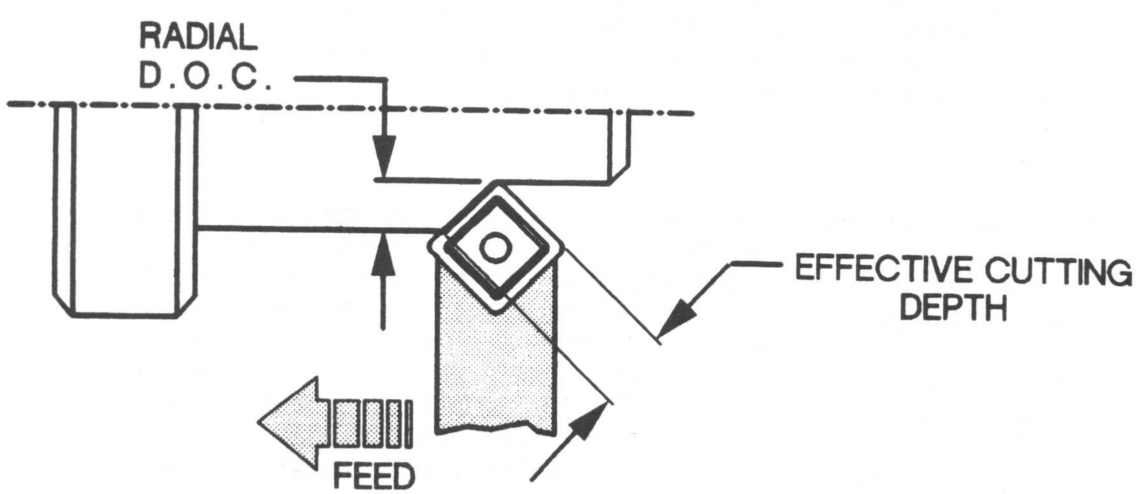
The lead angle of a turning tool is often selected based on the geometry of the workpiece or on the material condition. For example, when cutting through scale, interruptions or a hardened surface, a lead angled tool will allow you to maintain a reasonable rate of productivity without subjecting the insert edge to severe shock, thereby minimizing the effects of notching. This results by taking advantage of the chip thinning properties of the lead angle mentioned earlier.

This benefit, however, must be balanced against the possibility of increased part deflection or vibration due to the larger radial force created by the lead angle. The radial force increases as the lead angle of the holder becomes larger as stated earlier and illustrated by the picture below:



LEAD ANGLES AND CUTTING DEPTHS

Zero degree lead turning tools produce chips that are equal in width to the cutting depth of the turning operation. When a lead angle tool is introduced, the effective cutting depth and corresponding chip width will exceed the actual cutting depth of the workpiece. This phenomenon should be considered when using chip formation diagrams for specific chip breaker styles. See the pictures below for a complete explanation of the relationship between effective cutting depth and lead angles:



PRODUCTIVITY

Productivity in a turning operation can be improved by increasing the depth of cut, feed rate or cutting speed provided the appropriate level of machine horsepower is available. We will examine the effects of increased productivity on the cutting edge and workpiece as these parmaters are increased individually.

DEPTH OF CUT

The easiest cutting parameter to alter is the depth of cut. Doubling the depth of cut in a turning operation will double productivity without any increase in cutting temperature, cutting force per square inch of material removed (specific cutting force) or the tensile strength of the chip. The horsepower consumed will virtually double without any reduction in tool life (specific wear per inch of cutting edge length) assuming the cutting edge can withstand the added tangential cutting force. However, it is not always possible to increase the depth of cut to gain additional productivity, since there might not be any remaining material to remove.

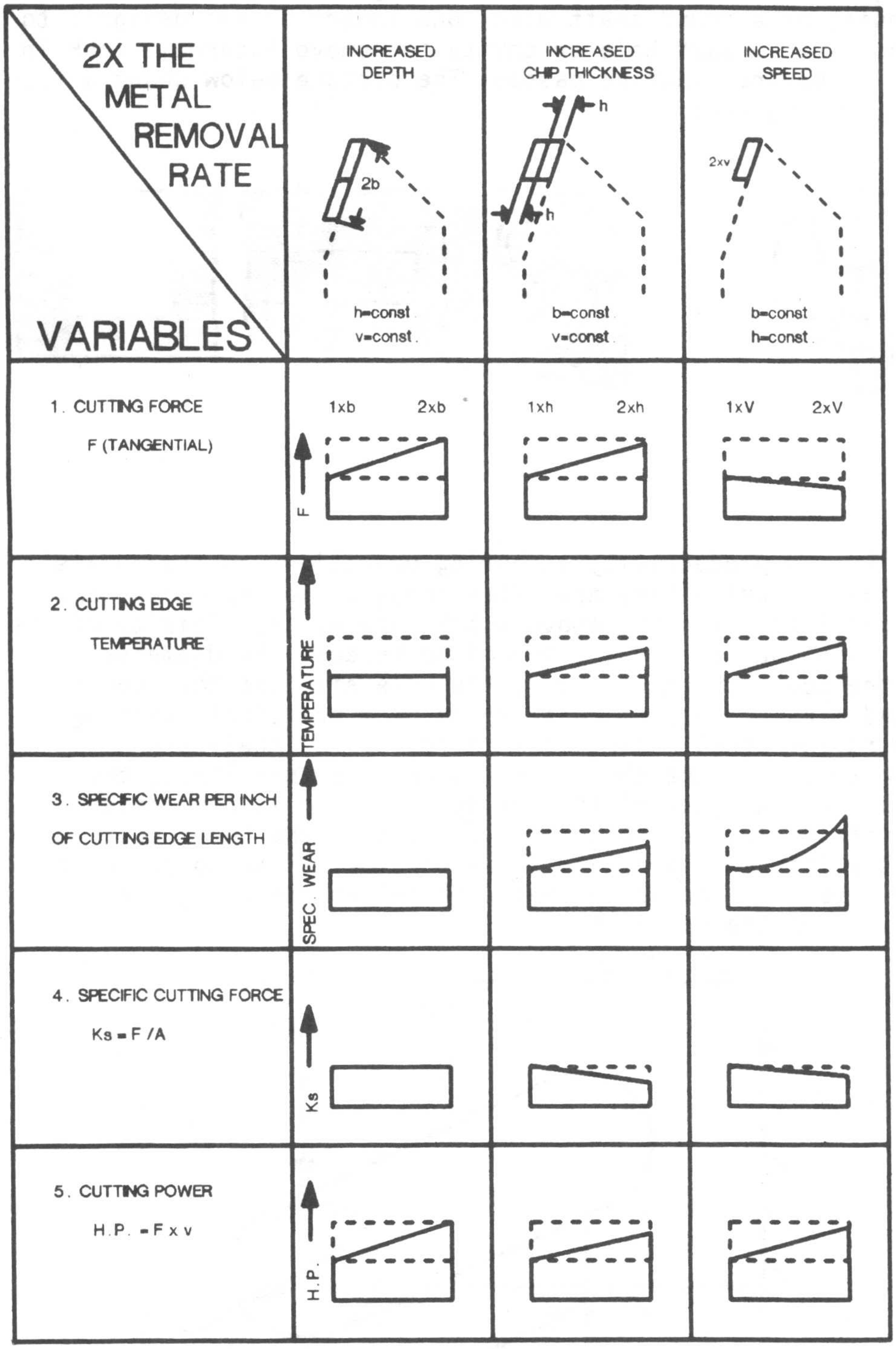
FEED RATE

The feed rate is often very simple to alter, therefore, it is the second most likely parameter to increase to gain added productivity. Doubling the feed rate makes the actual chip twice as thick thus making it much more difficult to curl and bend. However, the tangential cutting force, cutting temperature and horsepower increase, but they aren't doubled. This occurs because the tool is cutting more efficiently and less power is being wasted in heat generation per cubic inch of material removed. Tool life is reduced, but not halved. The additional force impeded on the cutting edge often causes cratering of the top rake insert surface due to the increased temperature and friction generated during the cut. But, the cutting force per square inch of material removed (specific cutting force) is actually decreased. Obviously, if the feed is increased without monitoring the effect on the tool catastrophic failure of the insert will result when the chip becomes stronger than the cutting edge.

SPEED

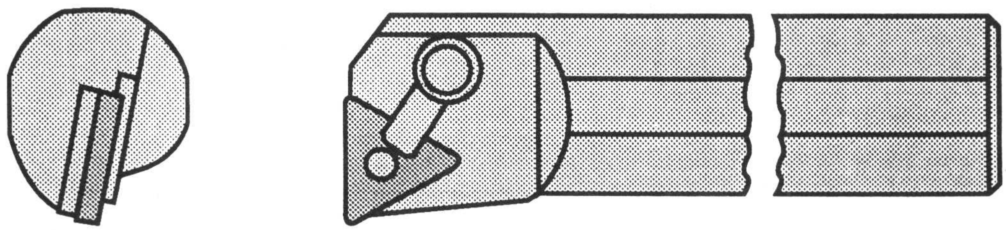
Increased cutting speed imposes numerous detrimental effects on the cutting edge. In addition to a significant rise in cutting temperature, doubling the cutting speed requires more power and imposes a substantial reduction in tool life (by more than half) in turning operations. The tangential and specific cutting forces actually diminish under this scenario as more power is consumed in heat generation per cubic inch of workpiece material turned. Therfore, the actual load on the top rake face of the insert is less, but cratering can still result due to the higher cutting temperature.

These three methods of increasing productivity are summarized in the chart below:

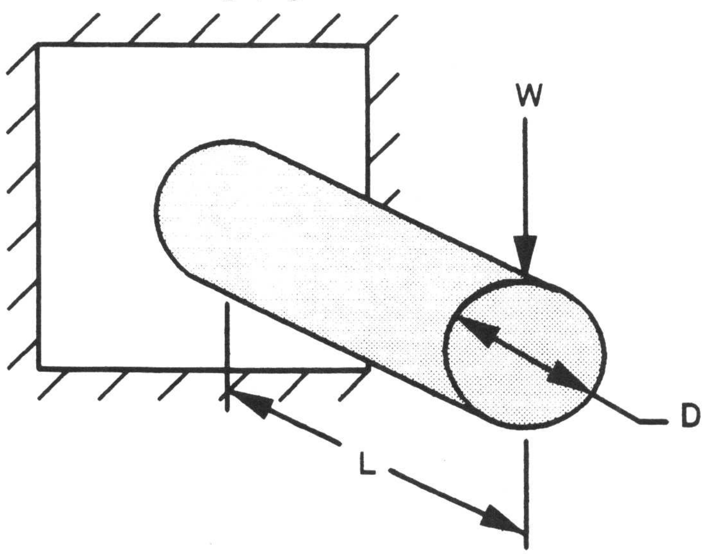


RIGIDITY IN BORING

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. The picture below shows a typical boring bar:



The key to productivity in boring operations is the tool's rigidity. Boring bars are often required to reach long distances into parts to remove stock. 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 a lack of rigidity and the increased possibility of vibration. If we consider a boring bar as a round beam we can approximate its deflection using the formula on the following page:



$$\text{DEFLECTION} = \frac{6.79 \times W \times L^3}{E \times D^4}$$

Where : D = The diameter of the beam in inches

E = The modulus of elasticity of the beam (PSI)

L = The length of the beam in inches

W = The load on the beam in pounds

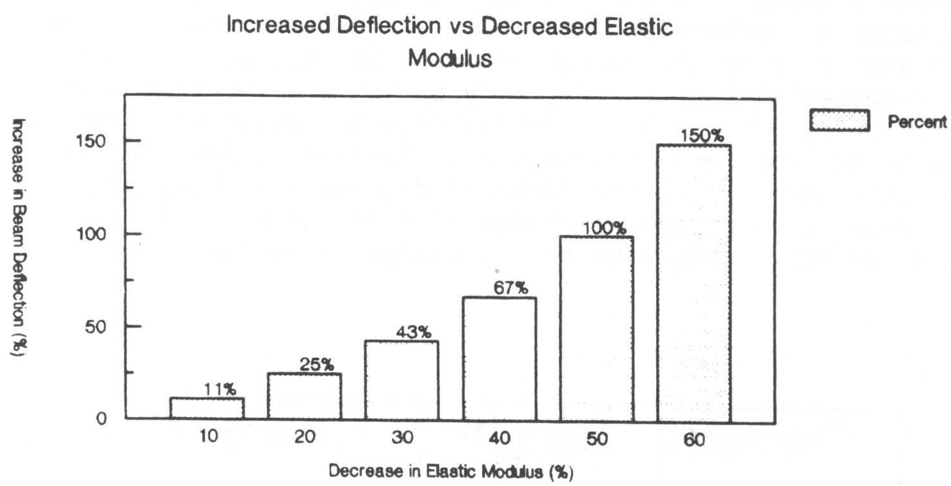
The purpose of discussing the beam formula is to illustrate some basic concepts, not to calculate specific deflections. Since one of the major keys to successfully applying boring tools is to provide the most rigid setup possible, it is important to understand what effects the stiffness of a boring bar and thus limits its deflection. The beam deflection can be minimized for a given load by altering either the beam length, diameter or material (modulus of elasticity). The beam diameter has the greatest effect on its stiffness or resistance to deflection. This is shown in the formula since the diameter is to the fourth power. A 15% decrease in the beam diameter will reduce rigidity by 48% and increase beam deflection by 92%, while a 15% increase in length will reduce rigidity by only 34% and increase deflection by 52%. Therefore, it's obvious that in terms of dimensional changes to a round beam (boring bar), a change in diameter will have a greater effect on rigidity than a change in length.

Diameter Change	Length Change	Change In Deflection
-15%	0	+92%
0	+15%	+52%
+15%	0	-43%
0	-15%	-38%

However, the beam material also plays an important role in its rate of deflection. This material characteristic is known as the modulus of elasticity **(E)** in the beam equation. The following list shows the modulus of elasticity for common boring bar materials:

MATERIAL	MODULUS (E)
Steel	30,000,000 psi
Heavy Metal	45,000,000 psi
Carbide	90,000,000 psi

A beam made of steel will deflect three times as much as one made from carbide, assuming they are identical dimensionally, since the modulus of elasticity for steel is one third the value of carbide. The modulus of elasticity is a material constant, characteristic of the material and independent of its hardness. Therefore, heat treating a steel boring bar will have no effect on its modulus of elasticity nor its resistance to deflection (stiffness). The effect of changing the material in a boring bar is illustrated by the graph below, where increased beam deflection is plotted against a decreasing modulus of elasticity:

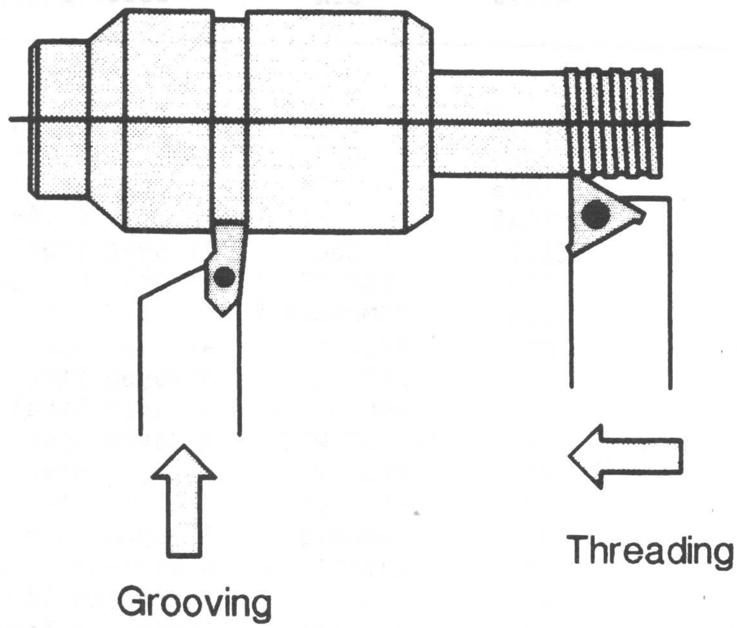


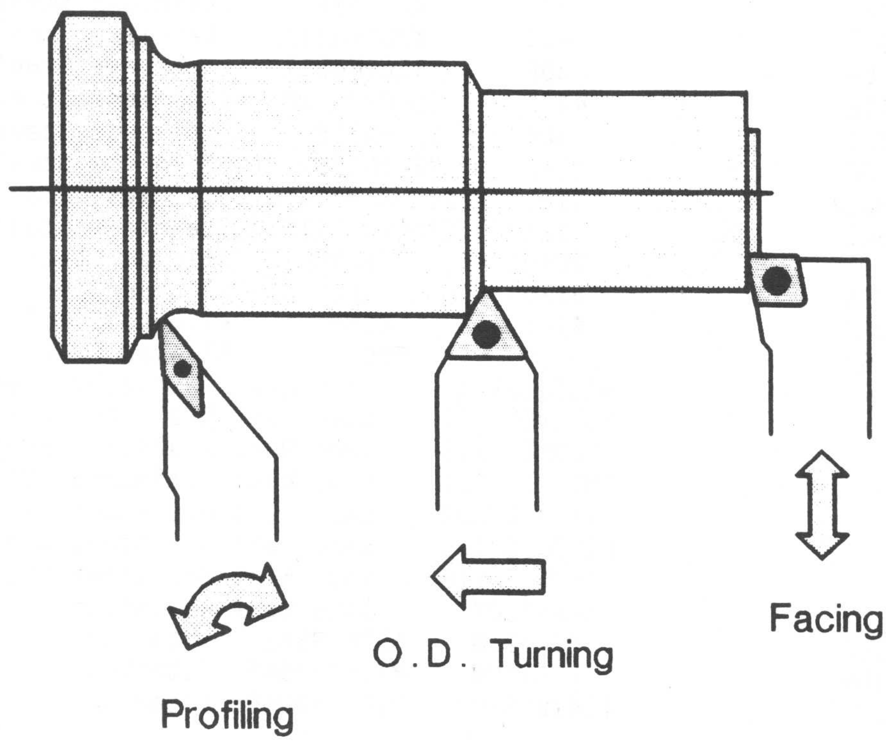
The key points to rigidity in boring applications are:

1. Changes in length will have less effect on rigidity and deflection than changes in the diameter of round beams or boring bars.
2. Changes in the beam or boring bar material will produce linear changes in rigidity and deflection.
3. Heat treating a steel beam or boring bar will not change its modulus of elasticity **(E)** and, therefore, its rigidity and rate of deflection will remain constant.

TURNING OPERATIONS

The following pictures illustrate the different types of single point external turning operations:





All turned workpieces will utilize one, a combination or all of these operations to produce a finished part. It is interesting to note that the operations shown above are all external turning operations, these same operations can be performed using boring bars on the internal diameters of workpieces.

INTERNATIONAL MATERIAL COMPARISON

Material Number	USA AISI#	Germany DIN	Description
1.0401	C1015	C15	Plain Stl. C < 0.2%
1.0402	C1020	C22	Plain Stl. C .2-.3%
1.0501	C1035	C35	Plain Stl. C .3-.4%
1.0503	C1045	C45	Plain Stl. C .4-.5%
1.0601	C1060	C60	Alloyed Steel
1.0715	1213	9SMn28	Plain Stl. C < 0.2%
1.0718	12L13	9SMnPb28	Plain Stl. C < 0.2%
1.2080	D3	X210Cr12	Alloyed Steel
1.2341	P4	X6CrMo4	Alloyed Steel
1.2344	H13	X40CrMoV51	Alloyed Steel
1.2436	D6	X210CrW12	Alloyed Steel
1.2550	S1	60WCrV7	Alloyed Steel
1.2601	D7	X165CrMoV12	Alloyed Steel
1.2842	01	90MnCrV8	Alloyed Steel
1.4006	410	(G)X10Cr13	Stainless Steels
1.4016	430	X8Cr17	Stainless Steels
1.4021	420	X20Cr13	Stainless Steels
1.4017	431	X20CrNi17	Stainless Steels
1.4104	430F	X12CrMoS17	Stainless Steels
1.4112	440B	X90CrMoY18	Stainless Steels
1.4301	304	X5CrNi189	Stainless Steels
1.4306	304L	X2CrNi189	Stainless Steels
1.4404	316L	X2CrNiMo1810	Stainless Steels
1.4541	321	X10CrNiTi189	Stainless Steels
1.6511	9840	36CrNiMo4	Alloyed Steel
1.7035	5135H	41Cr4	Alloyed Steel
1.7228	4147	50CrMo4	Alloyed Steel
1.8159	6150	50CrV4	Alloyed Steel
	HASTELOY X	same	Heat Resistant Steel
	INCONEL 625	same	Heat Resistant Steel
	INCONEL 718	same	Heat Resistant Steel
	INCONEL 901	same	Heat Resistant Steel
	NIMONIC 80A	same	Heat Resistant Steel
	NIMONIC 105	same	Heat Resistant Steel
	NIMONIC 118	same	Heat Resistant Steel
	WASPALOY	same	Heat Resistant Steel
3.7025	Ti 99.8	ASTM B381	Titanium
3.7164	TiAl6V4	MIL-T-9047	Titanium
3.7174	TiAl6V6Sn2	MIL-T-9046F	Titanium

HARDNESS COMPARISON TABLE

Tensile Strength (PSI)	(N/mm)	BHN	Rb	Rc
60,200	415	124	71.2	-
65,300	450	133	75.0	-
69,600	480	143	78.7	-
74,000	510	152	81.7	-
79,000	545	162	85.0	-
83,400	575	171	87.1	-
88,500	610	181	89.5	-
92,800	640	190	91.5	-
97,900	675	199	93.5	-
102,200	705	209	95.0	-
107,300	740	219	96.7	-
111,700	770	228	98.1	-
116,600	800	238	115.1	-
118,900	820	242		23.1
123,300	850	252		24.8
127,600	880	261		26.4
130,500	900	266		27.1
134,900	930	276		28.5
137,800	950	280		29.2
144,300	995	295		31.0
149,400	1030	304		32.2
153,700	1060	314		33.3
158,800	1095	323		34.4
163,100	1125	333		35.5
167,500	1155	342		36.6

Tensile Strength (PSI)	(N/mm)	BHN	Rc
172,600	1190	352	37.7
176,900	1220	361	38.8
182,000	1255	371	39.8
187,000	1290	380	40.8
191,400	1320	390	41.8
195,800	1350	399	42.7
200,800	1385	409	43.6
205,900	1420	418	44.5
211,000	1455	428	45.3
215,300	1485	437	46.1
220,400	1520	447	46.9
225,500	1555	456	47.7
231;300	1595	466	48.4
236,400	1630	475	49.1
241,400	1665	485	49.8
246,500	1700	494	50.5
252,300	1740	504	51.1
257,400	1775	513	51.7
262,500	1810	523	52.3
267,500	1845	532	53.0
272,500	1880	542	53.6
278,400	1920	551	54.1
283,500	1955	561	54.7
289,300	1995	570	55.2
294,400	2030	580	55.7

INDEX OF TERMS

Page

REVIEW QUESTIONS

1. Why would you select an insert with a chip breaker on just one side when several inserts are available with chip breakers on two sides?

2. What is top rake?

3. Why are side and end clearance angles required on turning tools?

4. What is the relationship between lead angle and the radial force generated during turning?

5. What is the cutting speed when turning a 4.5" finished diameter at 640 RPM?

6. Why is a square considered stronger than a triangular shaped insert?

7. If you wanted to increase productivity in a turning operation, why is increased feed preferred over increased speed?

8. How does a lead angle influence chip breaking?

9. How can you distinguish a right from left hand turning tool?

10. Why are hones, negative lands and chamfers used on inserts?

11. What has the greatest influence on the rigidity of a round tool, changes in length or diameter?

12. What factors influence chip breaking?

13. Which Hertel chip breaker would you use to finish turn a steel shaft at .005" IPR?

14. Why should feed rates be increased to provide adequate chip breaking when the depth of cut approaches the insert radius?

15. What are the four primary turning tools, and what types of operations are they used on?

TRAINING MANUAL

MILLING

INTRODUCTION

Milling is a complicated metal removal process which normally involves a multiple tooth rotating cutter body and a clamped workpiece fed in a linear direction against it. While turning is often used to create round surfaces with a single cutting edge, milling frequently uses several cutting edges in a single tool, to create flat or contoured surfaces. In milling, unlike turning, the cutting edge is cyclically entering and exiting the cut. This phenomenon alone adds to the complexity in application of the milling process and tools since it requires the machine operator or tool engineer to have an excellent working knowledge of cutter geometry, cutting edge density, horsepower and the rigidity required to effectively apply the process.

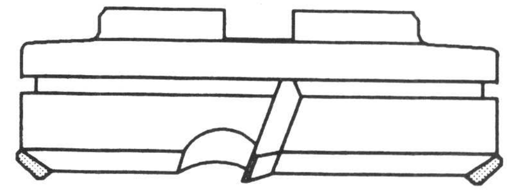
The range in use of the milling process is extensive in terms of variation in machining operations. For example, it is used to reduce the thickness of a 6 Ft. wide by 20 Ft. long ingot of aluminum by one inch in a single machine pass or to contour the radius on a titanium turbine blade. The versatility and efficiency of the milling process is best illustrated by the variety of industries in which it is commonly used including aircraft, automotive, power generation, oil field and machine tool manufacturing. Our focus for this material is to concentrate on milling in terms of modern indexable tooling with replaceable carbide inserts.

MILLING CUTTER STYLES

Milling cutters are generally defined by one of the three categories outlined below:

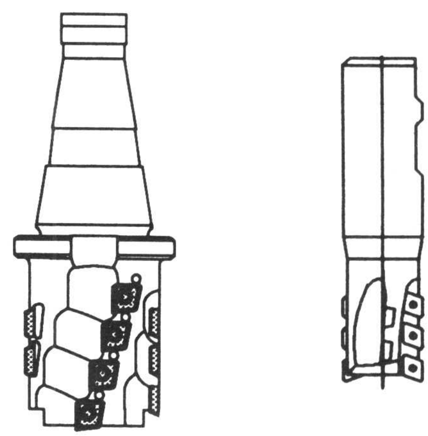
FACE MILLS

The face mill is normally defined as that milling tool in which one row of periphery mounted inserts is used to mill stock at a depth of cut equal to the amount of material removed from the workpiece perpendicular to the axis of rotation. Face mills are generally designed to remove large volumes of material in a single machine pass on a wide flat workpiece surface.



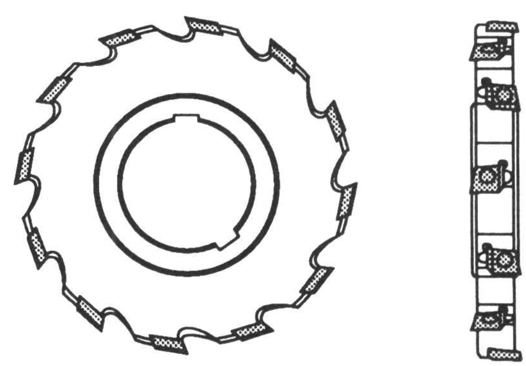
END MILLS

The end mill is normally defined as that milling tool in which the surface machined with the end of the cutter is perpendicular to the axis of rotation and the surface machined by the tool periphery is parallel to the axis of rotation. The cutting edges on an end mill are located on its face and periphery and extend along its length parallel to the axis of rotation. This milling tool is generally used to perform intricate operations where limited clearance prohibits the use of a face mill or where slab milling is required.



ARBOR OR SLOT MILLS

Arbor mills or slotting cutters are normally defined as those milling tools used to machine three surfaces simultaneously in a single machine pass. The slot produced using these tools has two sidewalls perpendicular to the axis of rotation and a bottom surface parallel to the axis of rotation. A slotting mill has cutting edges which mill on two faces and its periphery. Arbor mills are preferred over end mills, since they often are more rigid and productive on slotting operations.

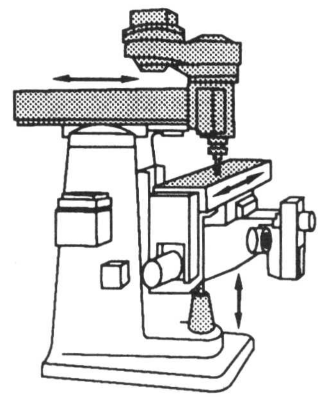


MILLING MACHINES

The CNC Machinery used to mill today's intricate workpieces is often more powerful and versatile than its manual counterparts of 10-15 years ago. However, there is still a significant amount of manual machinery in production applications. Let's take a look at the differences between these machines and their distinct features.

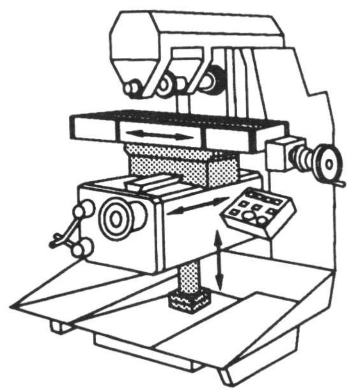
RAM TYPE MILLING MACHINES

This type of low horsepower (1.0-5.0) manual milling machine has been the most common machine for tool rooms in the past 30 years. The ram, table and knee of this simple machine have adjustment, thus providing several options for light duty machining with its vertical spindle.



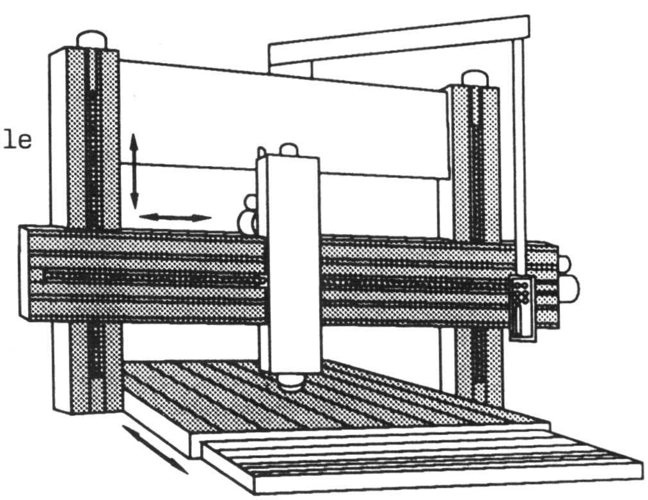
KNEE TYPE MILLING MACHINES

This manual machine has a cast column which holds the spindle fixed in a stationary position. The machine's saddle, knee and table are all adjustable under its horizontal or vertical spindle. The knee mill is a general purpose roughing machine for medium duty metal removal with spindle power from 5-50 horsepower.



PLANER TYPE MILLING MACHINES

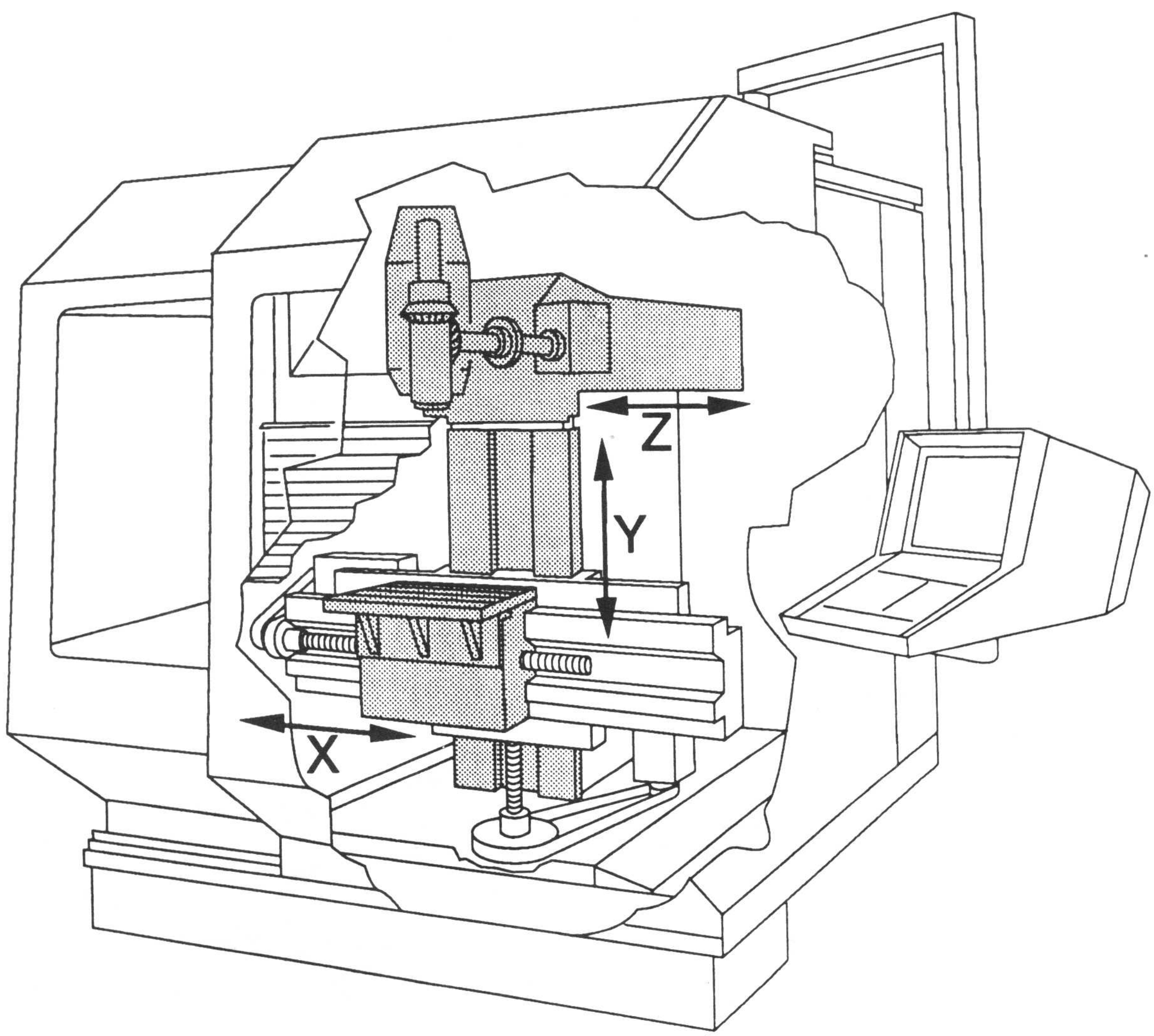
A planer mill may have a fixed bed or a combination of moveable heads and table instead of the fixed spindle arrangement common to the knee type machine. This style of machine is designed for heavy duty machining due to its rigid construction. Planer mills are often used to make long machine passes on workpieces which require a moderate number of simple machining operations. Planer mills have been built with up to 100 plus horsepower spindles.



The CNC machine tool is a product which is available in a multitude of configurations. The word "axis" refers to any rotational or linear machine move that can be controlled by a CNC command; this, however, does not include the spindle rotation which is never considered as an axis.

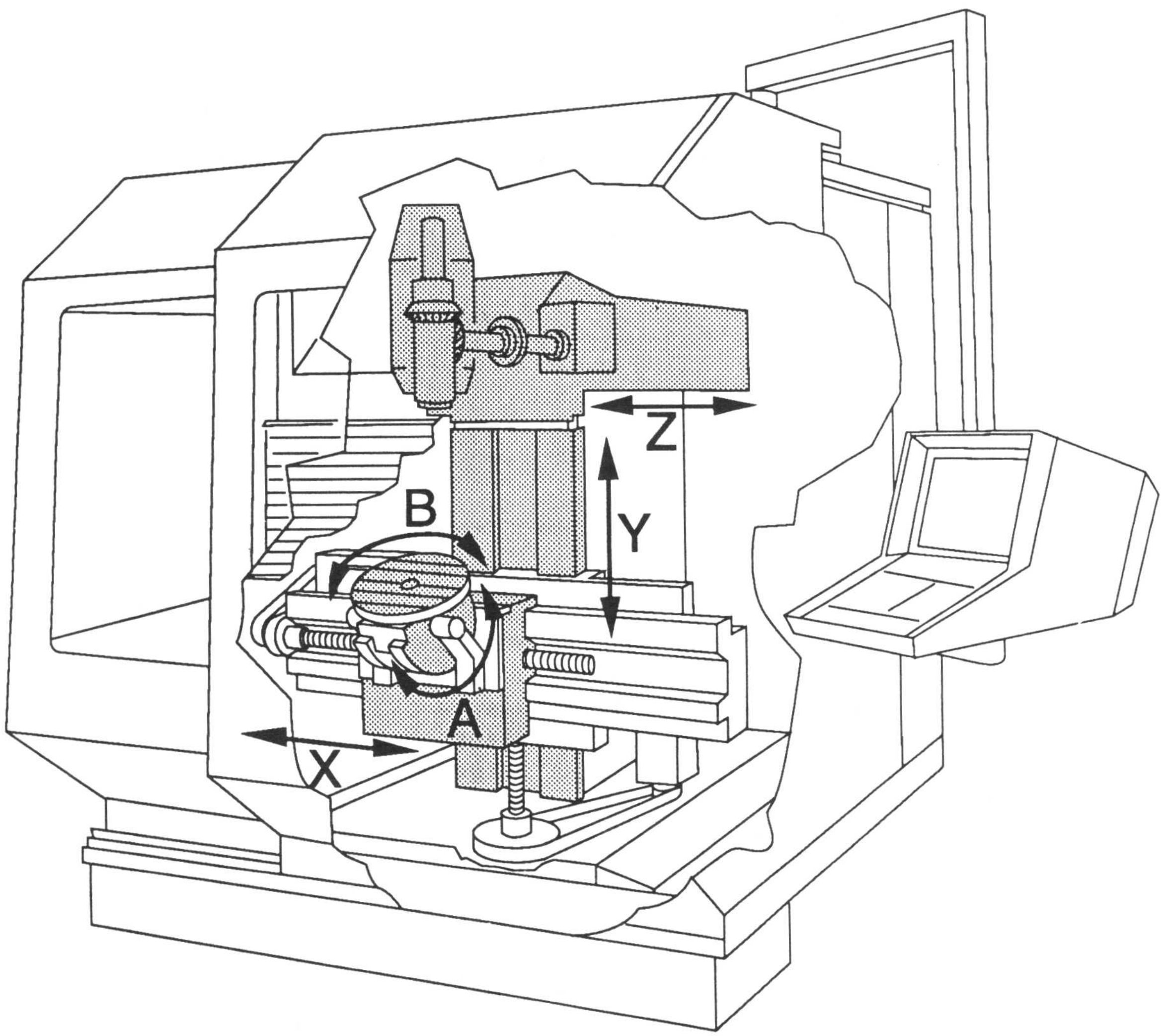
VERTICAL CNC MACHINING CENTER (3-AXIS)

The development of computer numerical controlled machinery (CNC) has led to the virtual standardization of a variety of machine tools. The vertical 3-Axis machining center can move in the x (front to back), y (right to left) and z (up and down) planes while under computer control. These machines are available in a broad range of workpiece cube (x,y, and z movement ranges) and horsepowers which are selected based on part requirements. The movement of the machine is accomplished in a variety of ways depending on the manufacturer. The head containing the spindle can be fixed and the machine table will move in all three axises x,y and z. There are three axis machines where the table is stationary and the spindle moves in the x,y and z planes.



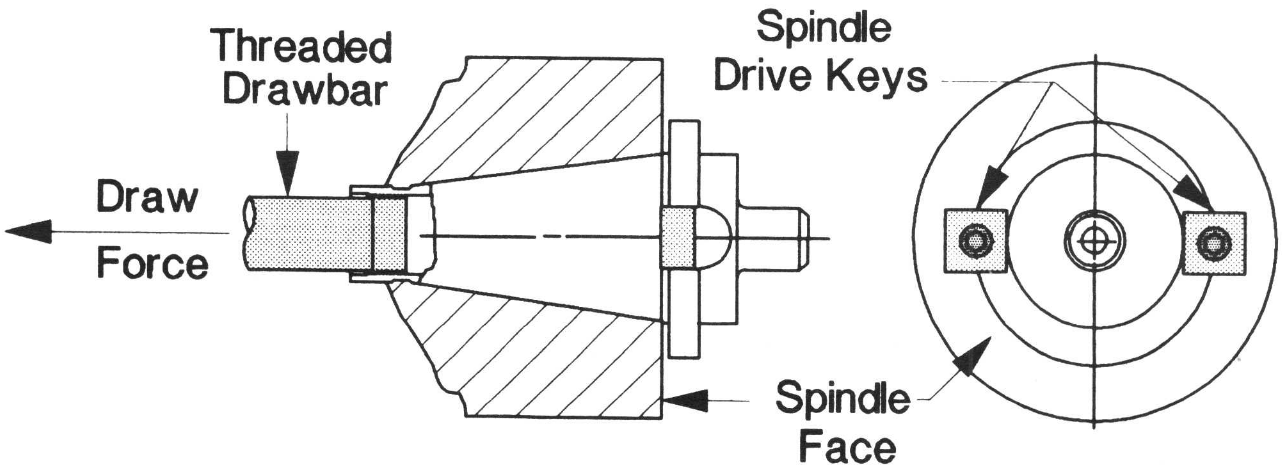
VERTICAL CNC MACHINING CENTER (5-AXIS)

The 5-axis machine tool normally has multifunctional tables which often provide two additional axes over the 3-axis machine. For example, if the stationary table on the 3 axis machine from the example above were replaced with a table that both rotated and tilted, we would have created a 5-axis machine tool. The x,y and z movements of the spindle would represent the first three axes, while the tilting and rotary movements of the machine table would account for the fourth and fifth axes. The movement of the machine can be provided at the spindle only, or at the table exclusively, or any combination of the two. There are grinding machines which have as many as 8-axes under CNC control, so the flexibility and complexity of the multiaxis machine tools are virtually without limitation.



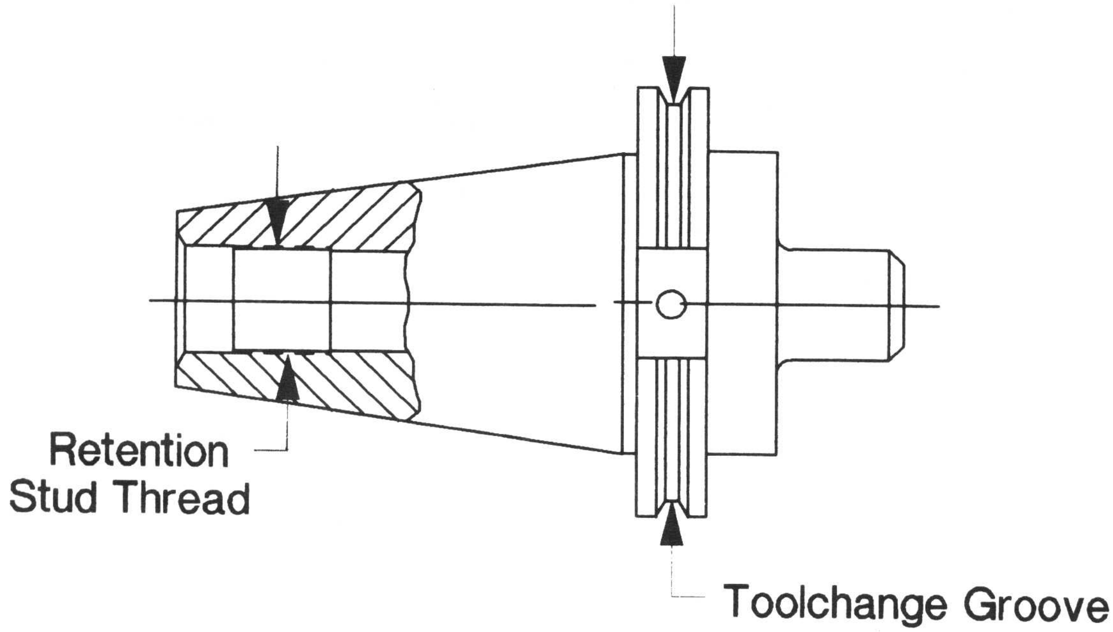
MACHINE SPINDLE ADAPTION

A face, end or slotting mill are rotating cutting tools which require a high degree of mounting accuracy and rigidity for optimal performance in terms of tool life and productivity. Therefore, the adaption of the tool to the machine spindle face is critical. This can be accomplished either by mounting the tool directly to the spindle face or by using an intermediate tapered adaptor. Many of the adaption systems used today are based on some type of tapered adaptor which is held in the machine spindle by either a threaded drawbar or a drawbar which grips the retention stud at the back of the adaptor. The adaptor is then driven by keys mounted on the spindle face. See the illustrations below:

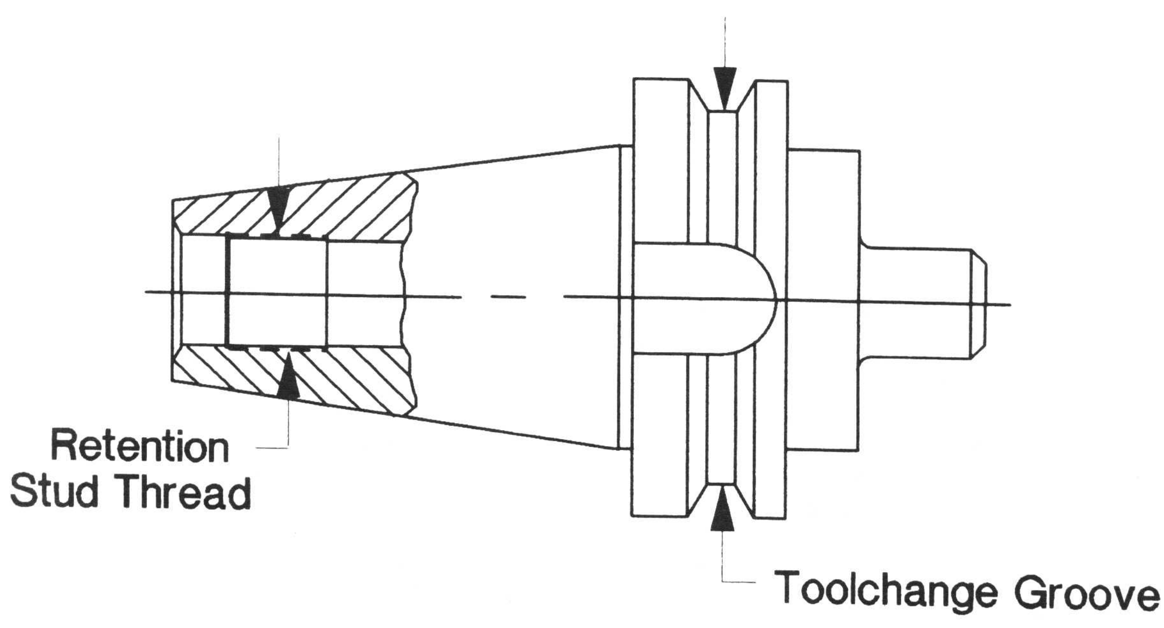


CATERPILLAR TAPERED SHANK ADAPTORS

The Caterpillar taper shank adaptor mounts directly into a machine spindles tapered hole and is secured by a draw bar which grasps the retention stud (knob) at the back of the adaptor. This system is available in a 30, 35, 40, 45 or 50 taper and is covered by ANSI Standard No. B5.50. The physical taper size is normally selected by the machine builder based on the available horsepower of the machine or the total length of the tools to be used on the machine. The unique feature of these holders is the design of the flange (V) which is used by the machine tool change device to pick the tool from the tool chain or magazine (storage system) and place it in the tapered spindle hole. The rear of the adaptor is threaded so a variety of retention studs can be used in a single holder to accommodate different machine grippers. This system is the American Standard. See page **48**:

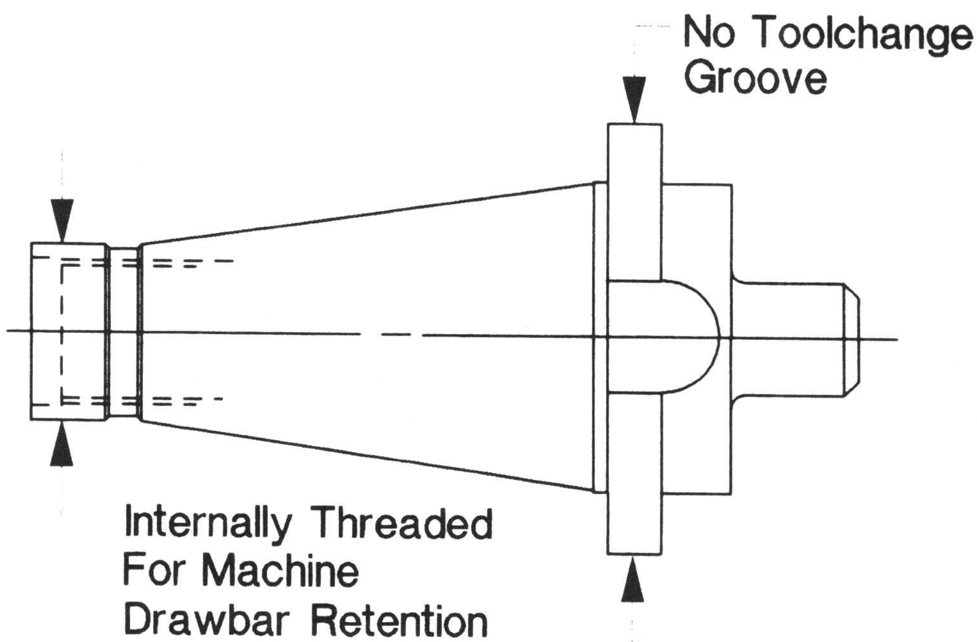


<u>BT TAPERED SHANK ADAPTORS</u>
This adaptor system has the same type of taper configuration as the CAT V system except the tool change gripper groove is different. This system also has a threaded section for interchanging retention studs (knobs). However, the threads on a BT adaptor are often metric and therefore, the retention studs are rarely interchangeable with those used in the CAT V system. The BT system works under the same principles as those described for the CAT V system except this system is the Japanese standard. See the illustration below:



NMTBA TAPERED SHANK ADAPTORS

The NMTBA (national machine tool builders assoc.) tapered adaption system has two distinct differences from the CAT V standard. First, instead of pulling the tool from a retention stud, the machine draw rod threads into the back of the adaptor to secure the tool axially in the machine. The second major difference is the fact that the system is used on machines without automatic toolchangers and therefore, the tool has no tool change groove, but has a flange ahead of the machine taper. See the illustration below:



FLAT BACK DRIVE

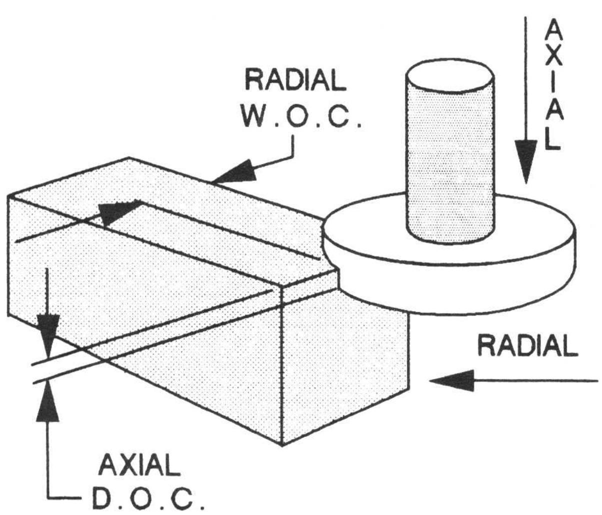
In some cases milling cutters can be mounted directly to the spindle face. When this occurs, the drive keys and threaded bolt mounting holes on the spindle match the keyslots and align with the bolt hole pattern on the cutter. The cutter is then mounted directly on the spindle face after inserting a centering plug in the spindle taper. This type of drive system is called flat back drive, and it provides the maximum rigidity possible since it has no extension or overhang from the spindle face. When the flat back drive of a face mill is different than the spindle design an adaptor or rotary toolholder is used to mount the cutter to the machine.

GEOMETRY OF THE CUT

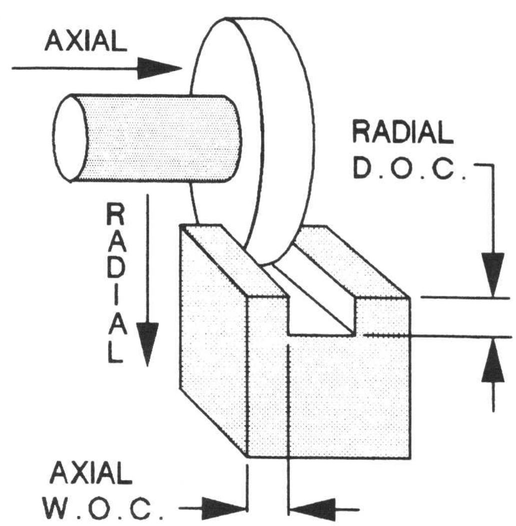
The milling process is a combination of rotary and linear movements which are used to remove excess workpiece material. The selection of the milling cutter style is often dependant on the part configuration and the machine type. For example, a slot can be produced using either an end mill or an arbor mill. In either case, the workpiece is fed linearly into a rotating tool, but in the case of an end mill the direction of the feed is radial while the feed direction would be tangential with an arbor mill. Although the direction of feed is different, in either case the tools axis of rotation is perpendicular to the feed direction. Arbor mills have periphery mounted inserts which stretch across the width of the tool while face mills possess a single row of periphery inserts mounted on the outer corner of the tool.

The distinction between face and arbor (periphery) milling can be made by establishing the direction of cut or stock removal. Face mills reduce the size of a workpiece by removing stock in an axial direction in line with the axis of rotation. Therefore, the depth of cut (D.O.C.) in facemilling is in an axial direction relative to the tool. Arbor mills, however, remove stock in a radial direction relative to the axis of rotation, and ,therefore, the depth of cut established using an arbor mill will always be a radial depth.

The width of material removed perpendicular to the axis of rotation in a facemilling operation establishes the radial width of cut (W.O.C.). In arbor milling, the width of cut is measured in line with the axis of rotation and therefore is the amount of stock removed from the workpiece in this direction. Please review the pictures below:



FACE MILL



ARBOR MILL

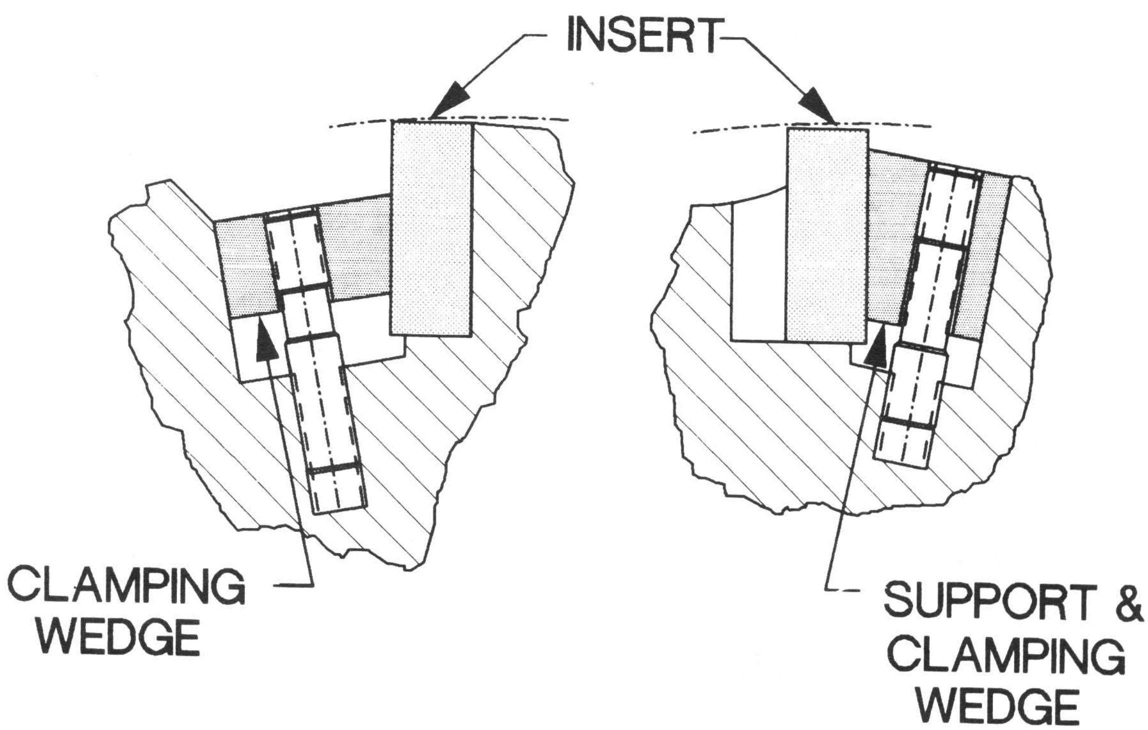
INSERT CLAMPING SYSTEMS

There are a variety of commercially available clamping systems for indexable inserts in milling cutter bodies. The examples shown cover the most popular systems now in use:

WEDGE CLAMPING

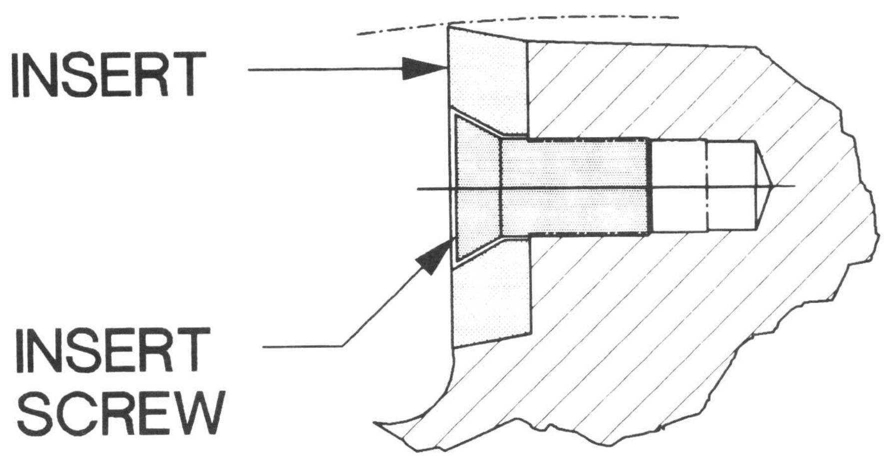
Milling inserts have been clamped using wedges for many years in the cutting tool industry. This principle is generally applied in one of the following ways: either the wedge is designed and oriented to support the insert as it is clamped or the wedge clamps on the cutting face of the insert forcing the insert against the milling body. When the wedge is used to support the insert, all of the force generated during the cut must be absorbed by the wedge. 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.

The wedge clamping 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 in the body. When this occurs, several of the other inserts already loaded in the milling cutter are removed and reset.



SCREW CLAMPING

This method of clamping is used in conjunction with an insert that has a pressed countersink or counterbore. A torx 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 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. Screw clamping is excellent for small diameter endmills 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.



CLAMP STUD CLAMPING

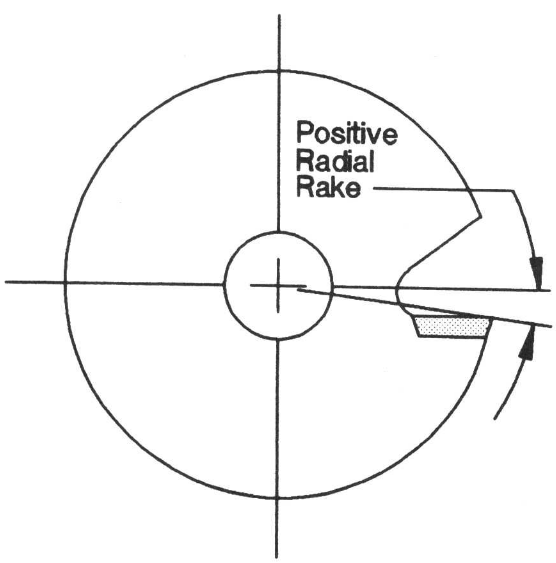
This clamping system utilizes a clamp stud (bent nail) and a screw to mount countersunk inserts in insert pockets with dovetail walls. This system is especially suitable for roughing operations since there are no protruding clamping elements to hamper chip flow since the clamping screw is not in direct contact with the insert, this system avoids the undesirable thermal effects which are common to the screw clamping system. The clamp stud pulls the insert down to the pocket floor and back against the pocket walls as the clamping screw is tightened. The unique pocket geometry and insert designs associated with this system (system Fix-Perfect) result in milling cutters designed to generate optimum cutting action and thus performance.

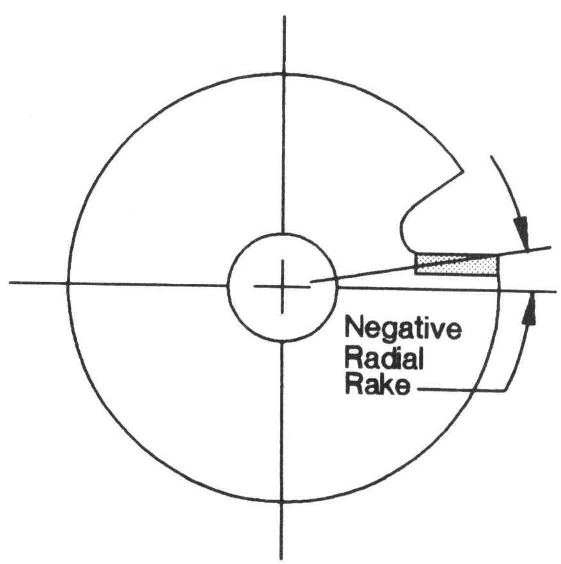
MILLING CUTTER GEOMETRY

The predominant angles most often used to describe milling cutter geometry are the Radial and Axial Rake and Lead. The peripheral Clearance and Bevel Angles are generally considered as being of secondary importance. The radial and axial rake in combination with the lead angle have a significant effect on the cutting edge entry into the workpiece, chip formation and flow, the tool life and the magnitude and direction of the resultant cutting force. Rake angles are normally described as being either positive or negative in orientation with respect to some reference. When the outer radial insert edge leads, the radial rake is positive. In the case of the axial rake, when the lowest point along the axial cutting edge leads, the axial rake is positive. Changes in the axial rake influence chip formation while alterations in the radial rake can effect the power consumption. Let's examine these different angles individually:

RADIAL RAKE ANGLE
When a face mill is mounted on a machine spindle and we look at the major diameter of the tool with its rotational axis as the center point, we are observing the tool in its radial view. In this view we can determine whether the radial rake is either positive or negative. If we extend a line from the center of the cutter to the outer most point on any cutting edge in this view, we can distinguish positive from negative radial rake. Positive radial rake is described in the picture below on the left, where the entire cutting edge is behind the radial line passing through the outer-most point (cutting diameter) on the insert.

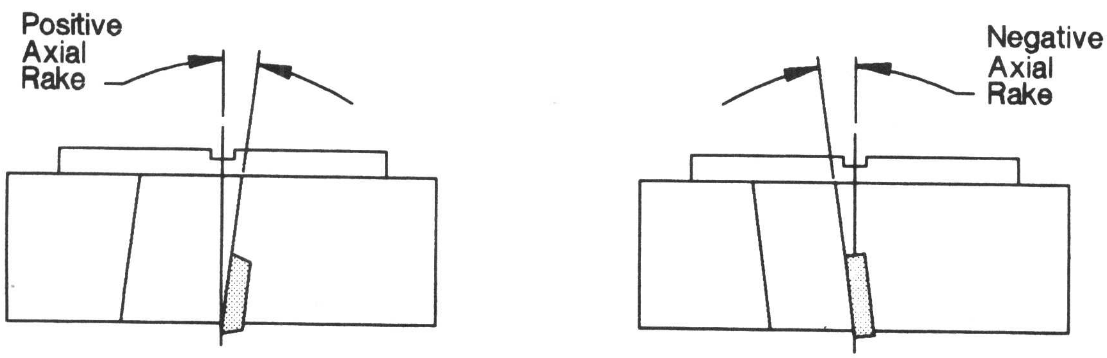
The right hand picture below illustrates negative radial rake, which occurs when the inside edge of the insert projects beyond the radial line passing through the outer most point (cutting diameter) on the insert.





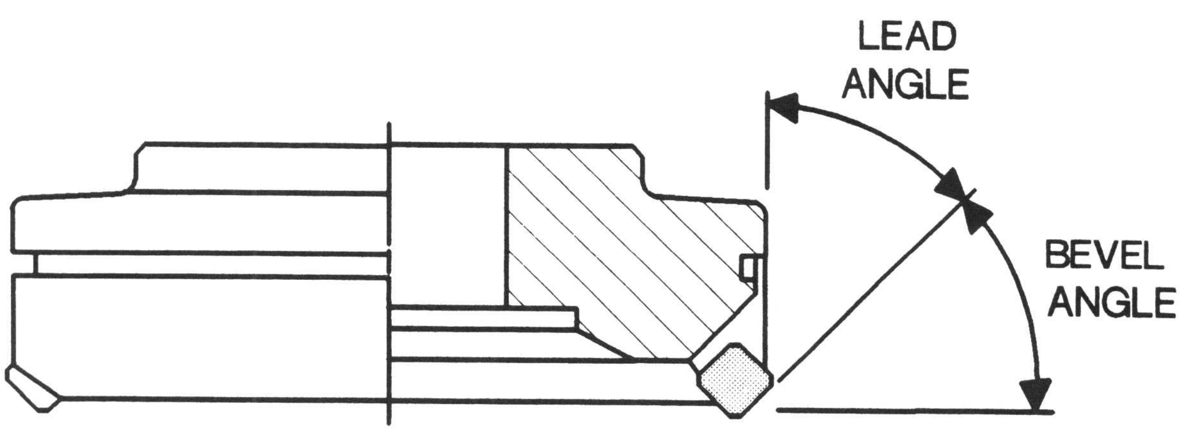
AXIAL RAKE

When we observe a face mill from the side with the mounting surface at the top of the picture and the thickness of the tool along its rotational axis, the face mill is in its axial view. If we extend the centerline of the face mill through the point on an insert which defines the cutter thickness, we can determine whether the axial rake is positive or negative. Positive axial rake is described by the picture below on the left, where the lowest point along the axial cutting edge will lead the insert into the cut. Negative axial rake is shown on the right in the picture, below where the lowest point along the axial cutting edge trails the remainder of the insert edge as it enters the cut.

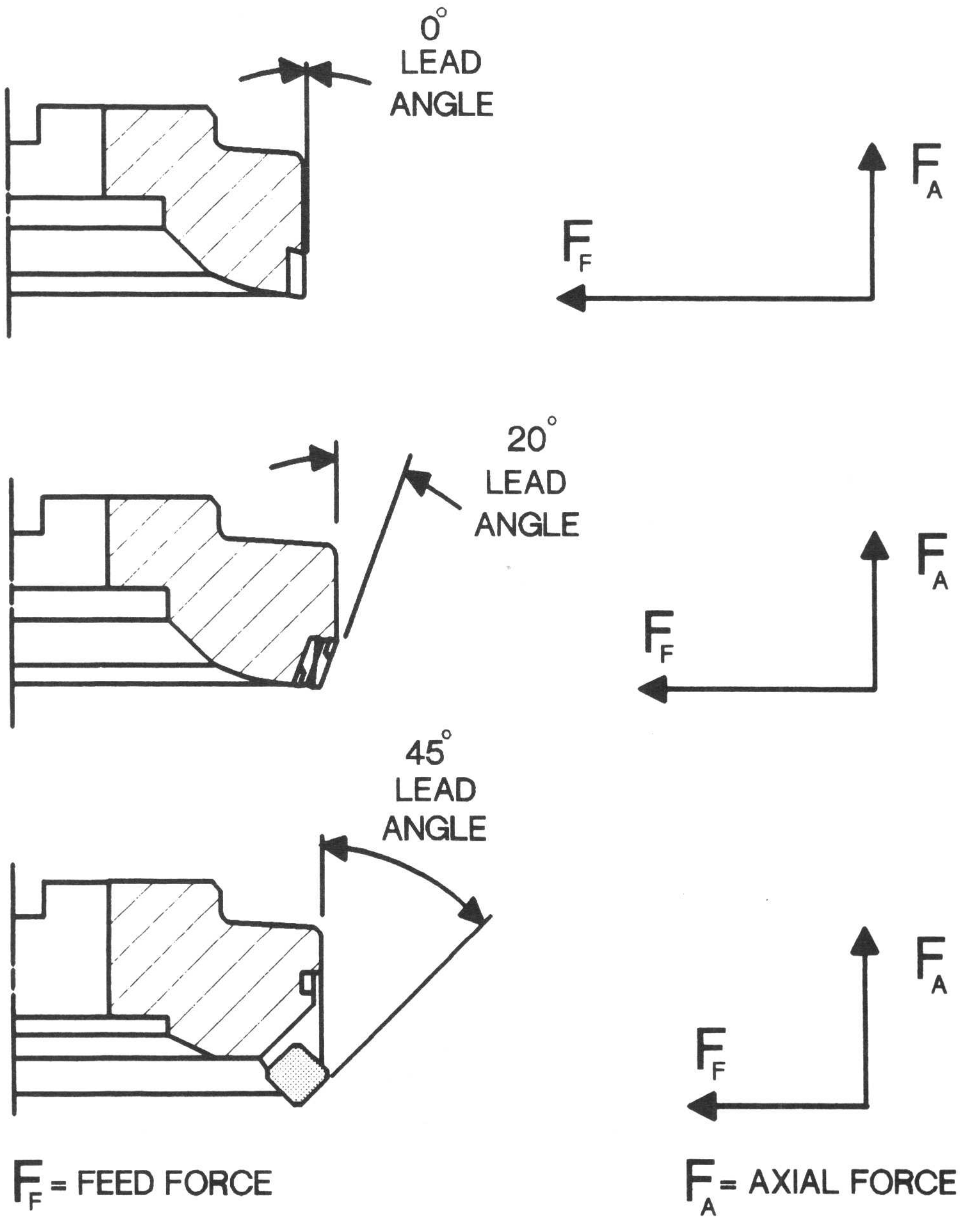


LEAD ANGLE

The lead angle is shown in the picture below and is defined as the angle which is described between a line drawn parallel to the cutter centerline through a point on the insert at the effective cutting diameter and a line projecting through the rake plane of the insert.



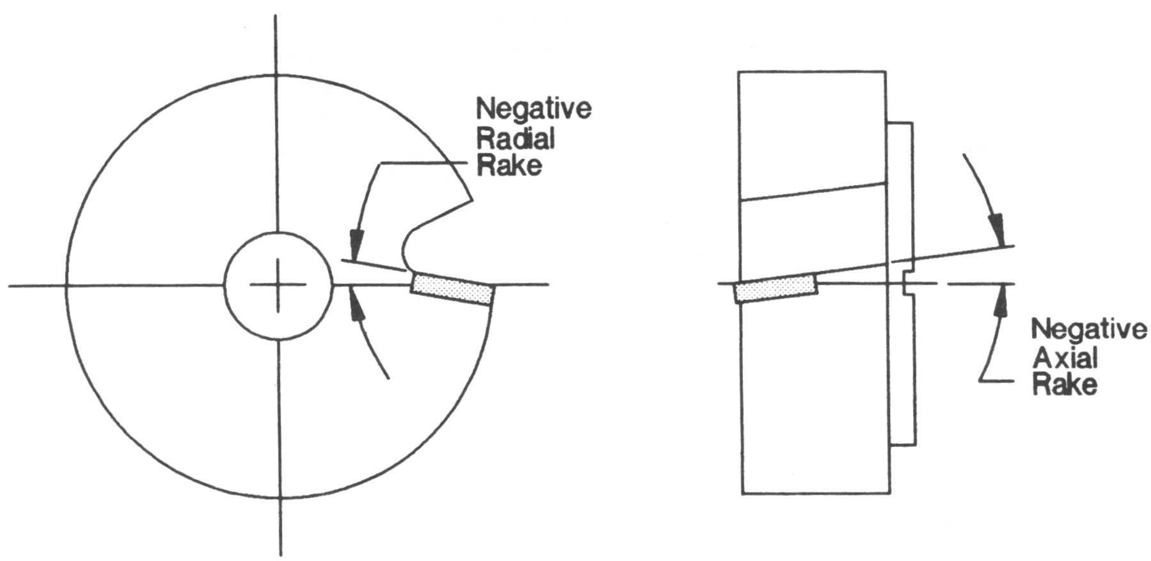
The complementary angle to the lead angle is called the bevel angle. Therefore, the total of these two angles for any given cutter is 90 degrees. The lead angle for ANSI standard milling cutters is normally 0, 15, 20, 30 or 45 degrees. As the lead angle increases for a given milling operation. the axial force component of the cut increases, but the actual chip thickness becomes a smaller percentage of the feed per insert. In high production face milling operations, large lead angle cutters are used to minimize breakout by taking advantage of the chip thinning effect of these tools. Milling tools with 0 degree lead angles generate higher feed forces and greater chip loads and, therefore, have a tendency toward creating vibration in the machining system. The pictures below show the effects of lead angles on the axial and feed forces.



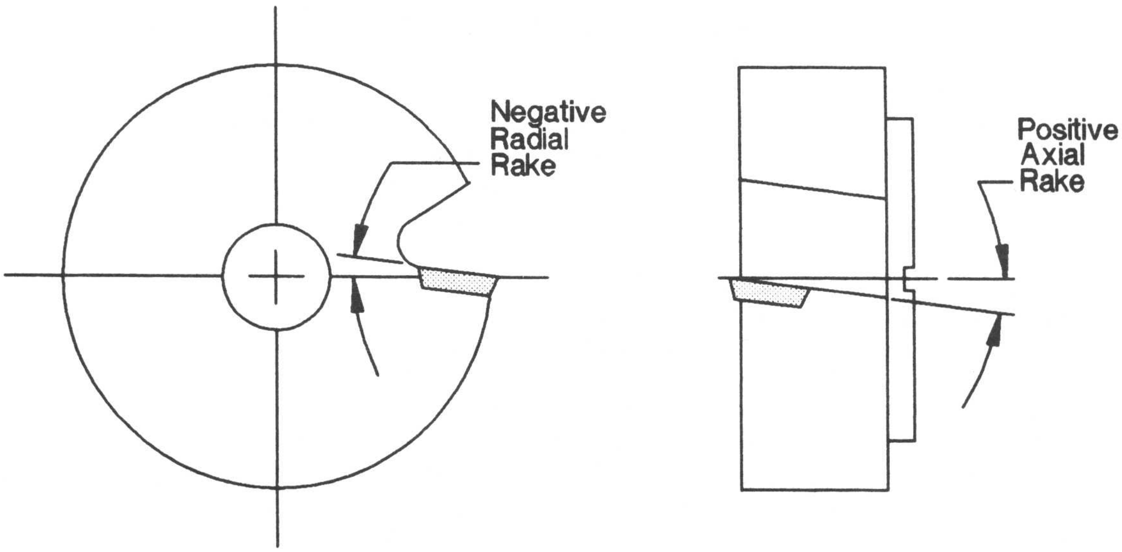
COMBINATIONS OF RAKE ANGLES

The four combinations of rake angles produce a variety of cutting actions which are favorable for some workpiece materials but detrimental to others. We will discuss these combinations of rake angles starting with the strongest geometry and progressing toward the weakest.

AXIAL AND RADIAL RAKE NEGATIVE
This combination provides the strongest insert geometries with the greatest number of available cutting edges or indexes. However, double negative geometries produce the highest cutting forces and, therefore, they generally require more rigid fixturing, parts and machine tools. This geometry is very popular in high production applications for roughing of grey cast iron components. These tools are often used on harder workpiece materials where a strong insert crossection is required to survive the extreme impact loading and stress delivered by hardened steels and cast iron. However, this is a poor choice in geometry for deep wide cuts on long chipping ductile materials, since the chip normally welds and crushes against the cutting edge and will not flow smoothly away from the cutting zone, thereby making it difficult to evacuate and hence reducing insert life.

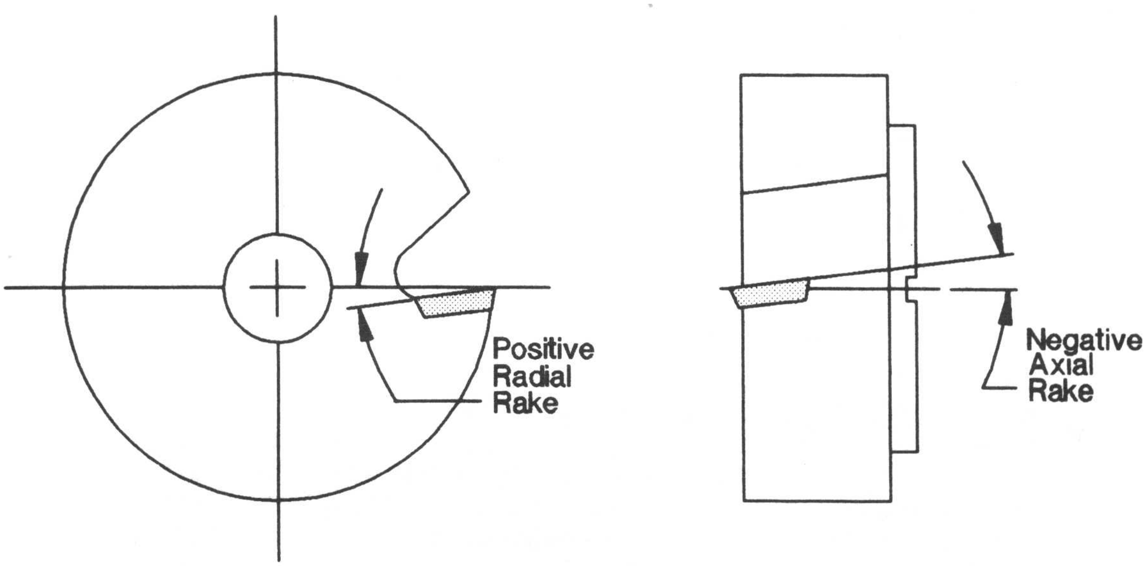


AXIAL RAKE POSITIVE-RADIAL RAKE NEGATIVE
This geometry produces spiral chips which are easily evacuated in deep wide cuts on long chipping workpieces while also taking advantage of a very strong stable cutting edge. The cutting geometry of this angle combination is well suited to low carbon steels, die steels and a wide variety of ductile ferrous materials.

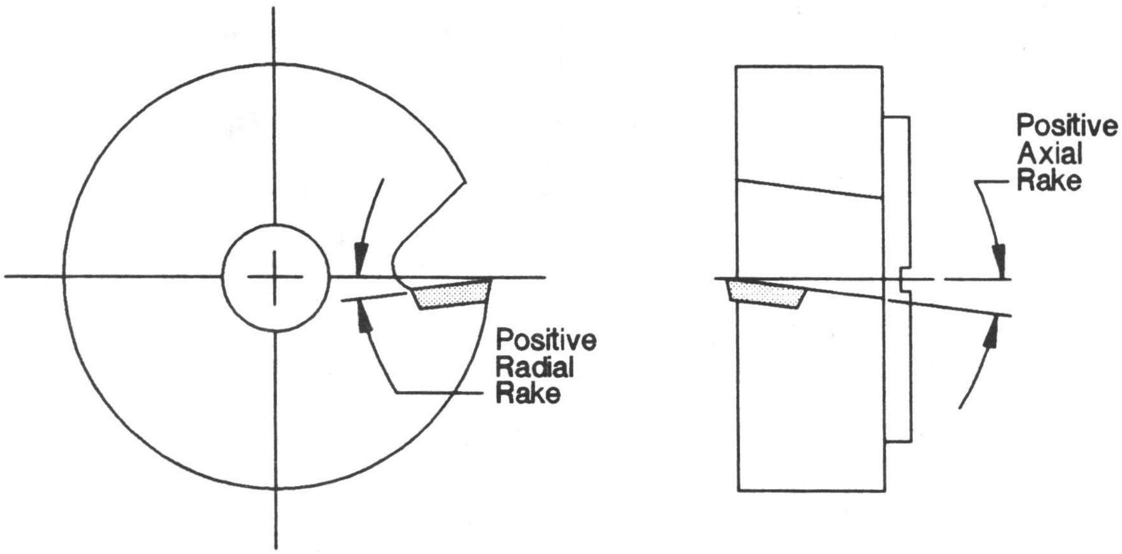


AXIAL RAKE NEGATIVE-RADIAL RAKE POSITIVE

The **"Fix-Perfect"** cutting system is designed around this combination of angles. When this geometry is combined with the clamp stud insert mounting system, high productivity milling is achieved. The negative axial rake protects the insert corner from the impact normally experienced at the entry of the cut, since the upper portion of the insert contacts the workpiece first. The positive radial rake provides low cutting forces and power consumption, making this a very efficient cutting geometry on steels and cast irons. The angle combination produces spiral chips which are easily evacuated on long chipping workpiece materials.



AXIAL AND RADIAL RAKE POSITIVE
This angular combination produces the lowest cutting forces but it also uses the weakest most fragile cutting edge geometry. Productivity on ferrous workpieces is low with double positive geometry in comparison to other angle combinations. This geometry produces long spiral chips on long-chipping materials. Double positive geometry is often used on light metals such as low silicon aluminums, magnesium etc.

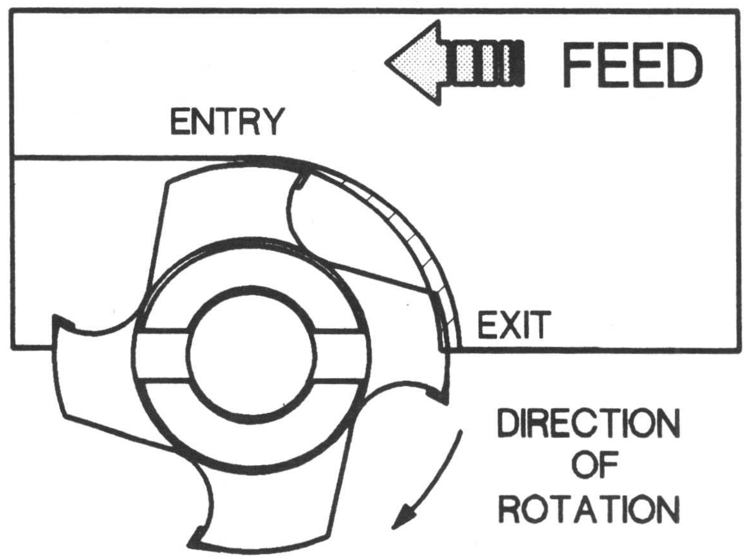


CONVENTIONAL AND CLIMB MILLING

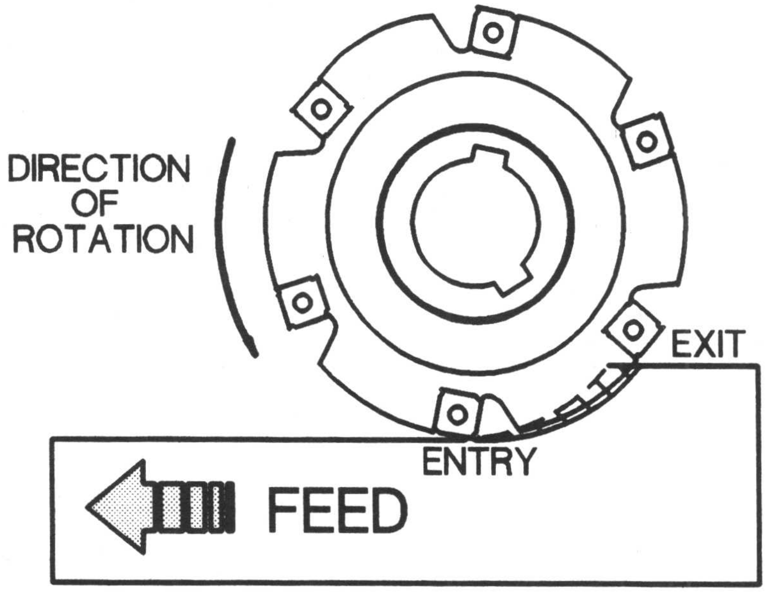
The application of the milling tool in terms of its machining direction is critical to the performance and tool life of the entire operation. The two options in milling direction are described as either conventional or climb milling.

CONVENTIONAL MILLING
Conventional or up-milling occurs when the cutter rotation pushes against the direction of the workpiece feed. The insert enters the workpiece at zero chip thickness and exits the cut at a maximum chip thickness.

CONVEN. MILLING (FACE MILL)

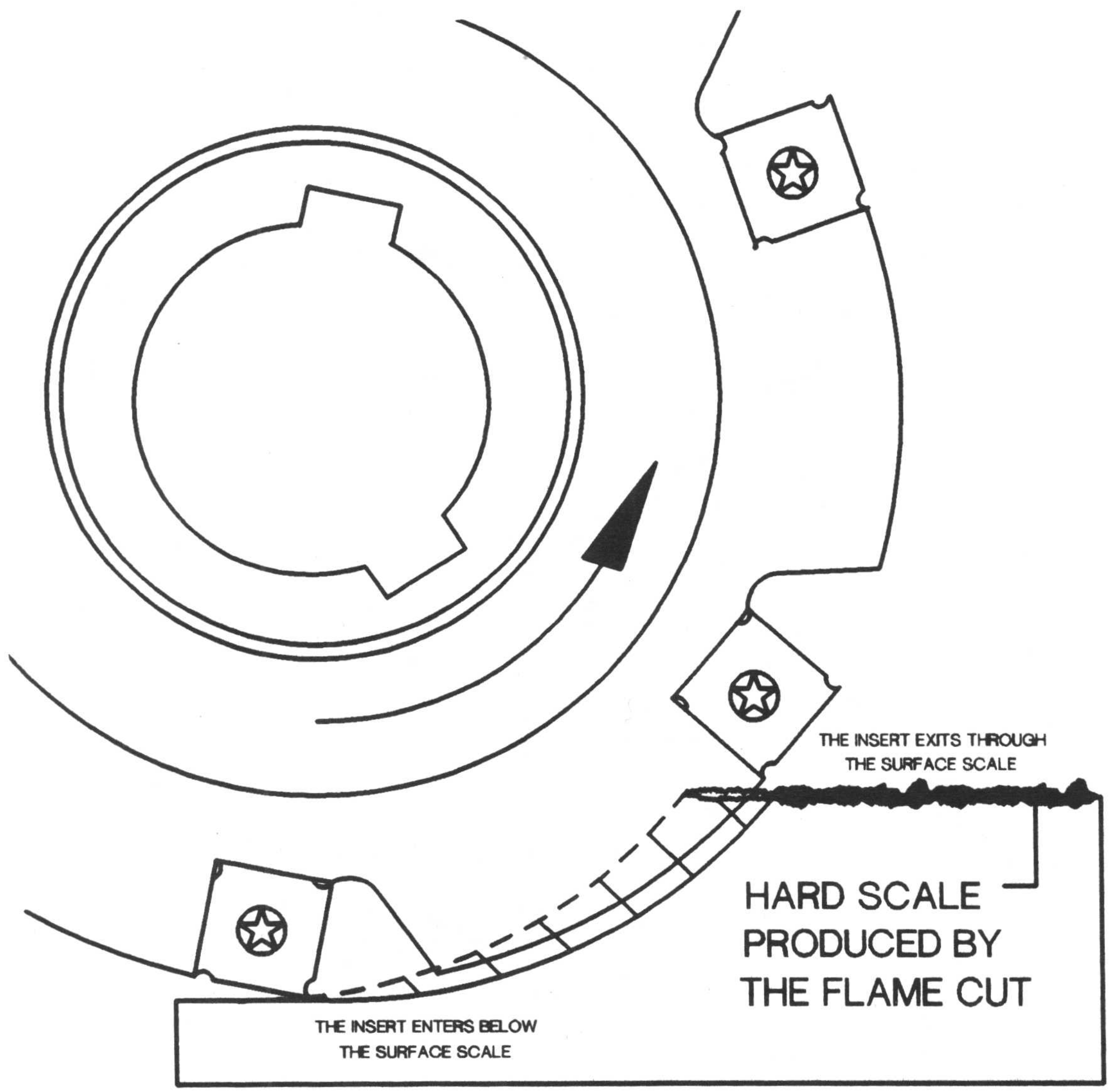


UP MILLING (ARBOR MILL)



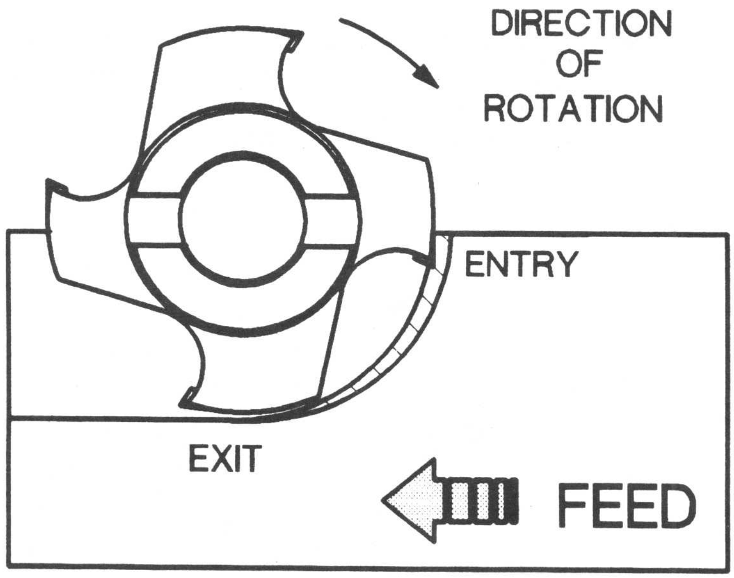
Therefore, the insert enters and slides until a minimum chip thickness is attained to produce chip formation. This phenomenon hardens the surface of the part and thus each successive insert has to work its way through the hardened layer which ultimately reduces tool life. Conventional milling is undesirable in those cases where chip welding is a problem since if a chip is welded to an insert and carried around to the entry point of the cut it will be wedged between the workpiece and the insert, resulting in breakage.

Up-milling is recommended for materials with a hard scale or flame cut surface. Using this method, the insert enters below the part surface and thus avoids any increased impact loads which could be detrimental to tool life.

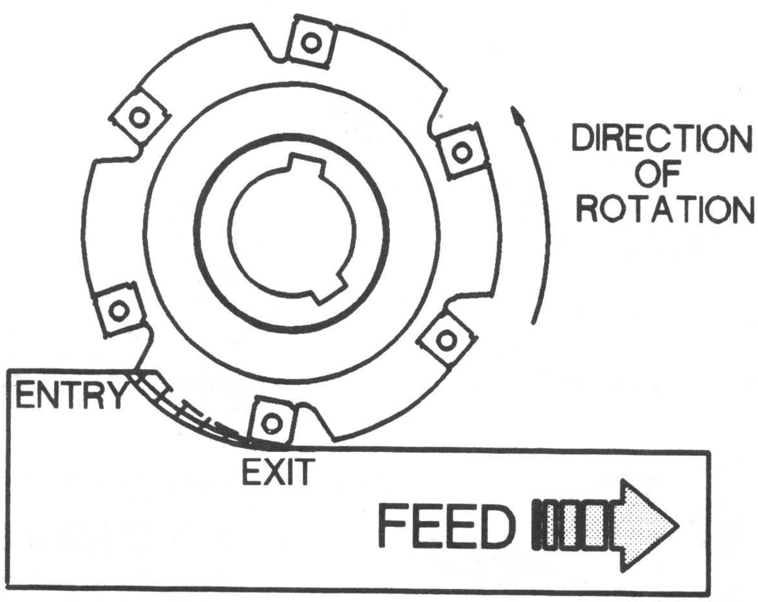


CLIMB MILLING
Climb or down-milling occurs when the cutter rotation pulls in the same direction as the workpiece feed. The insert enters the workpiece at a near maximum chip thickness and exits the cut at zero chip thickness. This is true whether the milling cutter selected is a face or an arbor mill.

CLIMB MILLING (FACE MILL)



DOWN MILLING (ARBOR MILL)



Climb milling is preferred over conventional milling because it causes less heat and requires less feed drive power which reduces overall power consumption for the operation. Since there is no sliding action at the entry of the cut, climb milling produces better tool life than conventional milling.

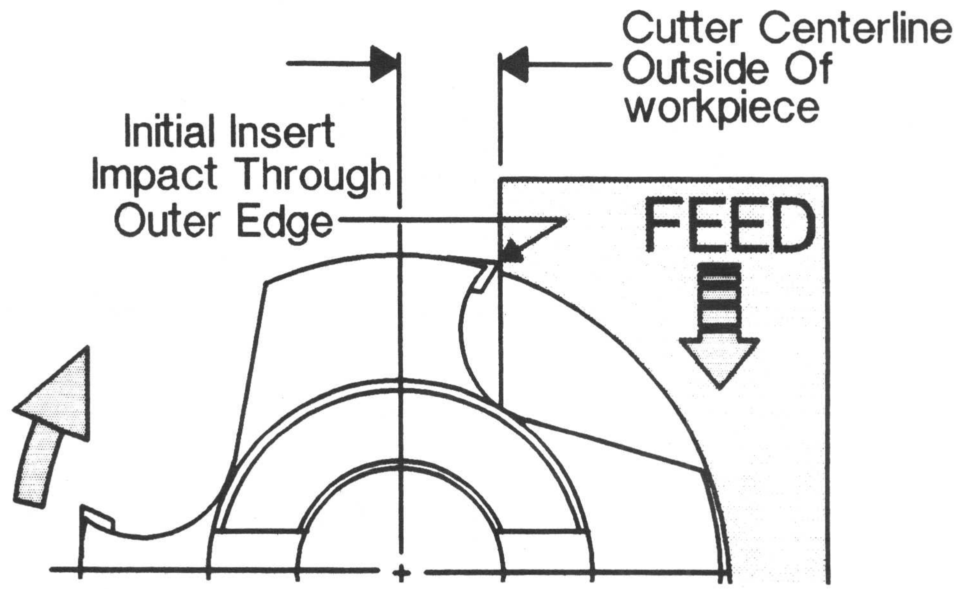
Climb milling should be avoided only when the feed drive system has backlash which will cause momentary fluctuations in the actual feed per insert and insert breakage due to increased chip thickness. Climb milling is the preferred method for high-nickel alloys, titanium and stainless steels to avoid the disastrous effects of workpiece workhardening on tool life.

MILLING DIRECTION

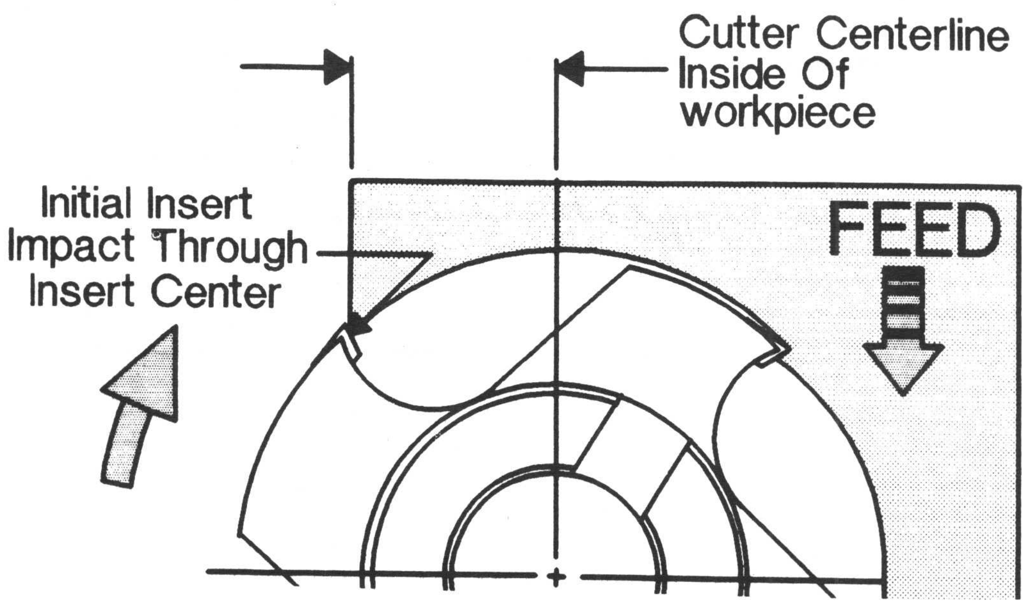
CLIMB	CUTTING PARAMETER	CONV.
<	HEAT GENERATION	>
<	POWER CONSUMPTION	>
>	EFFICIENCY	<

MILLING CUTTER POSITIONING

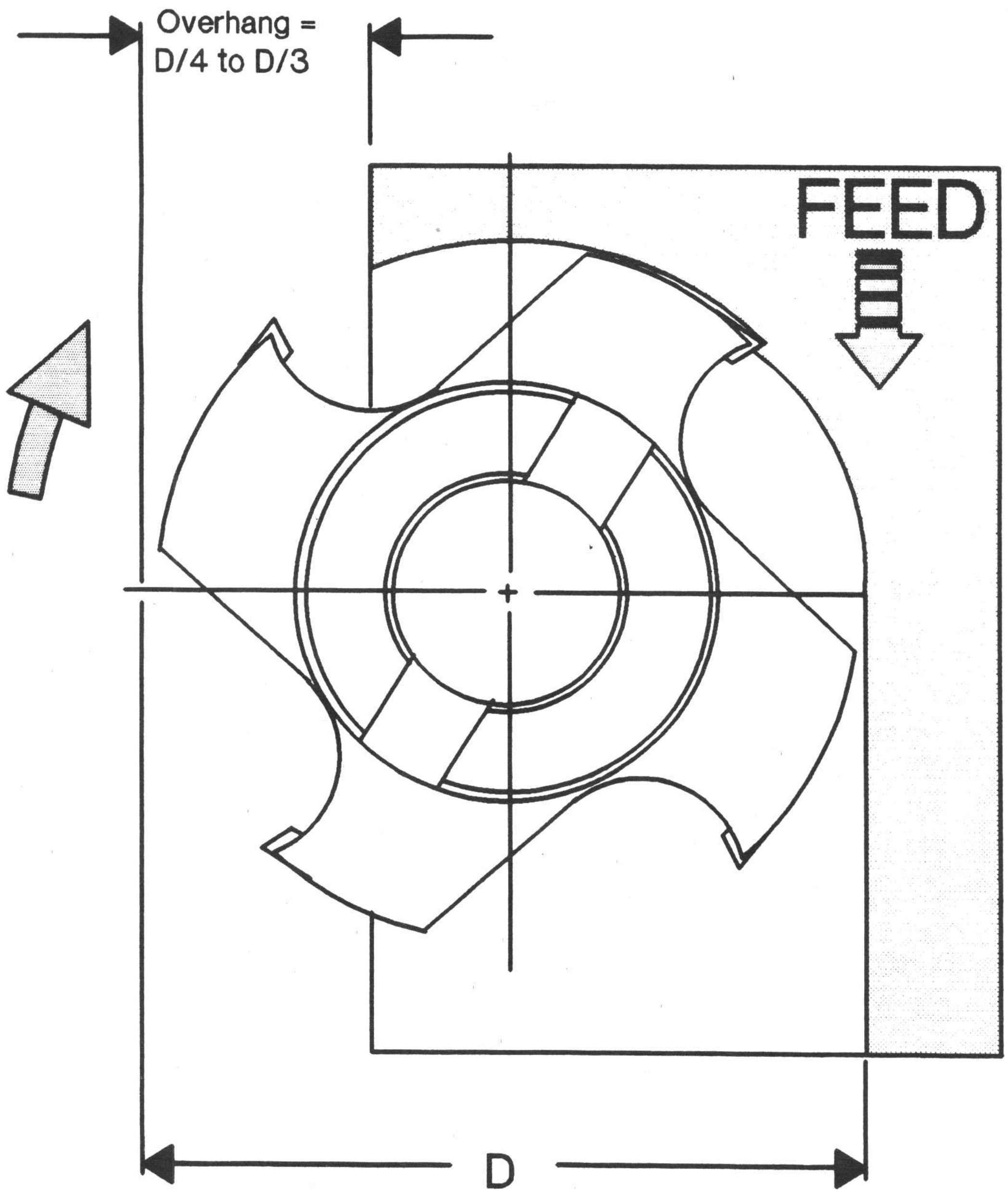
In milling, the position of the cutter relative to the workpiece can influence the level of vibration and the ultimate tool life of the operation. Milling inserts are subjected to impact shock loading every time they enter the cut. The severity and magnitude of these loads can be controlled by properly selecting the appropriate cutter geometry, diameter and position which best suits the machining condition.



In the picture above, the initial impact of the cut is absorbed by the outer edge of the insert, since the centerline of the cutter is outside the workpiece. In this case the impact shock load is entirely concentrated on the most vulnerable point on the insert which often leads to fracture or premature wear through micro-chipping.



The picture on the bottom of page 26 illustrates proper cutter positioning with the centerline of the cutter inside the workpiece and the entry impact load transmitted through the center of the insert. As a general guideline, 1/4 to 1/3 of the cutter body should overhang the workpiece on the entry side of the cut to insure that the appropriate entry angle is maintained. This is illustrated in the picture shown below:



CALCULATING CUTTING SPEEDS AND FEEDS

The measure of how easily a workpiece material can be machined is often called machinability. This measure is a comparison of the actual time it takes to machine a volume of a particular material, and, therefore, it is used to compare the relative machining rates of one material versus another (i.e. 1045 steel versus 6Al-4V titanium). Machinability is dependant upon the cutting speed and hence the ultimate feed rate that is attained to remove the appropriate volume of material required to satisfy the test conditions.

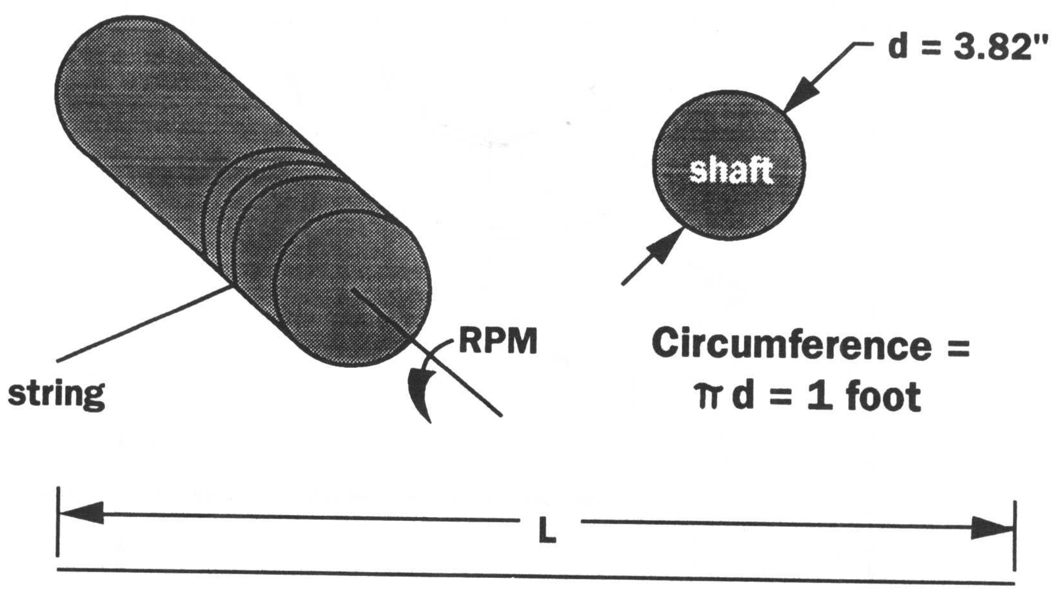
Cutting speed is defined as the distance in feet that is traveled by a point on the cutter periphery in one minute. This can easily be illustrated by imagining the length of string (in feet) that would wrap around a rotating shaft in one minute. If we select a shaft of 3.82" in diameter, the periphery or circumference of the shaft would equal 1 FT., since the circumference of something round is equal to:

$$\text{CIRCUMFERENCE} = \pi \times (d)$$

Where :

π = a constant of 3.1416

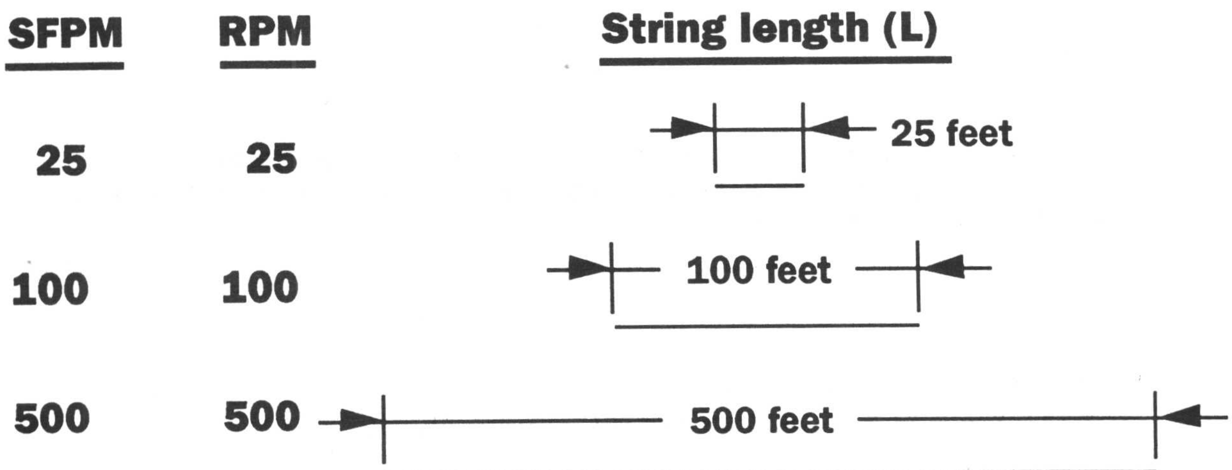
d = diameter in feet



L = (1 foot) x (RPM)

L = The length of string wrapped in one minute

Therefore, a 3.82" diameter shaft is .318 feet in diameter, and when we multiply the shaft diameter in feet by π , the result is a shaft circumference of 1 foot. In this simple example, if we rotate the shaft at 50 RPM (revolutions per minute) the shaft periphery would travel 50 Feet each minute and 50 feet of string (L) would have wrapped around the shaft diameter, or in other words, we would have attained a speed of 50 SFPM (surface feet per minute). If the same test was repeated at 500 RPM, the shaft periphery would travel 500 feet in one minute and 500 feet of string (L) will have wrapped around the shaft diameter, or we would have attained a peripheral speed of 500 SFPM.



The formula normally used to calculate cutting speed is as follows:

SFPM = (Cutter Circumference) x (RPM)

Where :

SFPM = surface feet per minute, or the distance traveled by a point on the cutter periphery each minute

Cutter Circumference = the distance around the cutter periphery in feet

RPM = revolutions per minute

In the case of a cutter, the circumference is:

Cutter Circumference = $\pi/12 \times (d) = .262 \times d$

Where :

Cutter Circumference = the distance around the cutter periphery in feet

π = is a constant, of 3.1416

d = the cutter diameter in inches

By substituting for the cutter circumference, the cutting speed can now be written as:

$$\mathbf{SFPM = .262 \times d \times RPM}$$

This formula can be used to determine the cutting speed at the periphery of any rotating tool, or in the case of turning at the outer workpiece diameter.

EXAMPLE #1

What surface speed would you be running if you milled a mild steel casting with a 4" diameter cutter at 525 RPM?

$$SFPM = .262 \times d \times RPM = .262 \times 4'' \times 525 = 550$$

ANSWER : SFPM = 550

EXAMPLE #2

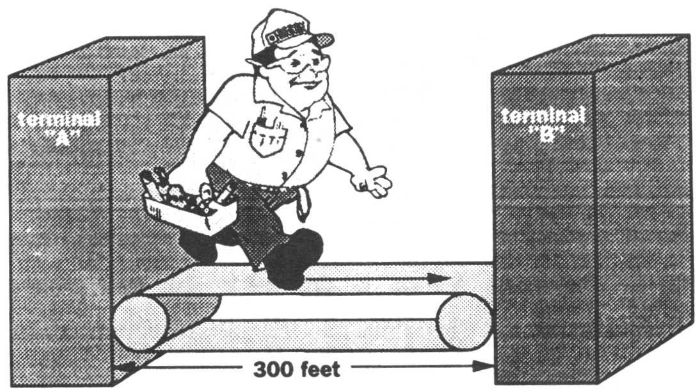
What RPM would you select to maintain a cutting speed of 4000 SFPM on a low silicon aircraft aluminum using a milling cutter with an 8" diameter?

$$RPM = \frac{SFPM}{.262 \times d} = \frac{4000}{.262 \times 8} = 1908$$

ANSWER : RPM = 1908

Once the cutting speed is established for a particular work-piece 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.

This concept is best explained by considering the time it takes for a person to travel between two airline terminals. If terminal "A" is 300 Ft. from terminal "B", and it takes 3 minutes to ride a moving sidewalk from "A" to "B", than we could say the sidewalk is transferring people at the rate of 100 Ft./minute (FPM) or 1200 inches/minute (IPM). Please review the following simple cases shown below:



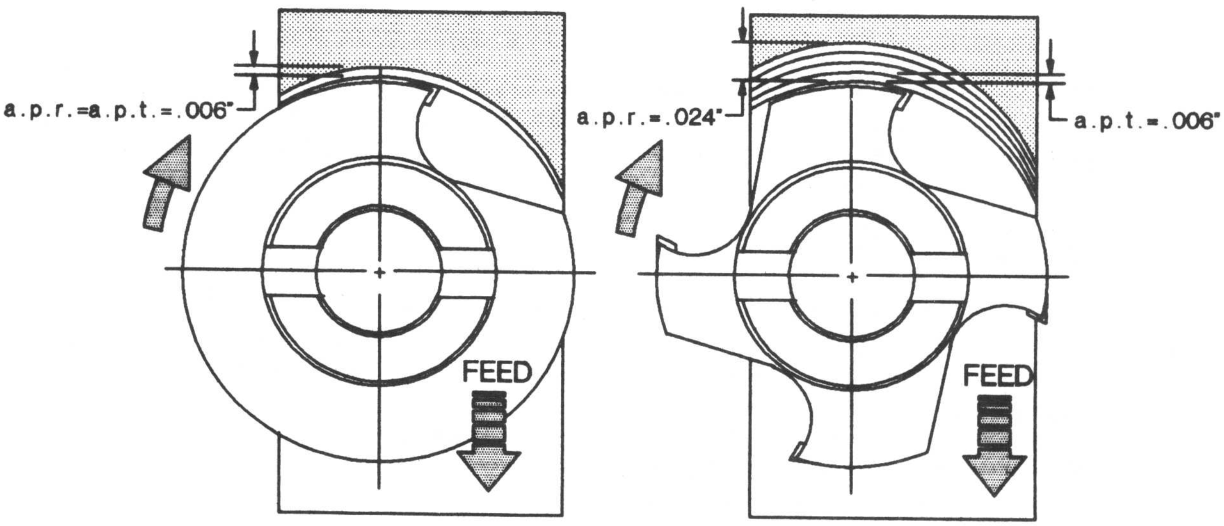
Time leaving Terminal "a" = 12:00 P.M.

Time arriving Terminal "b" = 12:03 P.M.

$$\text{Feedrate} = \frac{\text{300 feet}}{\text{3 minutes}} = \text{100 Ft. / Min.} = \text{1200 In. / Min.}$$

time of travel	distance	resulting feedrate
3 minutes	300 Feet	100 Ft./Min. = 1200 In./Min.
6 Minutes	300 Feet	50 Ft./Min. = 600 In./Min.
10 Minutes	300 Feet	30 Ft./Min. = 360 In./Min.

When we establish feed rates for milling cutters, the goal is to attain the greatest 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 appro-priate feed rate for a specific milling application, the RPM, number of effective inserts (N) and feed per insert in inches (IPT or a.p.t.) should be supplied. The concept of feed per insert is illustrated by the pictures on the following page.



The milling cutter shown above in the left hand picture (1 insert) will advance .006" 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), a.p.t. (advance per tooth) and an a.p.r. (advance per revolution) of .006". The same style of cutter with 4 inserts is shown above in the right hand picture. However, to maintain an equal load on each insert, the milling cutter will now advance .024" at the centerline every time it rotates one full revolution. The milling cutter on the right is said to have an IPT and a.p.t. of .006", but an a.p.r. (advance per revolution) of .024" (.006" for each insert). These concepts are used to determine the actual feed rate of a milling cutter in IPM (inches per minute) using one of the following formulas:

IPM = (IPT) x (N) x (RPM)

or

IPM = (a.p.t.) x (N) x (RPM)

where:

IPM = inches per minute
N = # of effective inserts
IPT = inches per tooth
a.p.t. = advance per tooth
RPM = revolutions per minute

EXAMPLE #3
If you were milling automotive grey cast iron using a 3" diameter face mill with 6 inserts at 400 SFPM and 30.5 IPM, what a.p.r. and a.p.t. would this be?

$$\text{RPM} = \frac{\text{SFPM}}{.262 \times d} = \frac{400}{.262 \times 3} = 509$$

$$\text{a.p.r.} = \frac{\text{IPM}}{\text{RPM}} = \frac{30.5}{509} = .060''$$

$$\text{a.p.t.} = \frac{\text{a.p.r.}}{N} = \frac{.060}{6} = .010''$$

ANSWER : a.p.r. = .060"

a.p.t. = .010"

EXAMPLE #4
If you were milling a 300M steel landing gear with a 4" diameter 45 degree lead face mill (containing 8 inserts) at 320 SFPM and a .006" advance per tooth, what feed rate should you run in IPM?

$$\text{RPM} = \frac{\text{SFPM}}{.262 \times d} = \frac{320}{.262 \times 4} = 305$$

$$\text{IPM} = \text{a.p.t.} \times N \times \text{RPM} = .006 \times 8 \times 305 = 14.6$$

ANSWER : IPM = 14.6

The following basic list of formulas can be used to determine IPM , RPM, a.p.t., a.p.r. or N depending on what information is supplied for a specific milling application:

IPM = inches per minute

N = number of effective inserts

a.p.r. = inches of cutter advance every revolution

a.p.t. = inches of cutter advance for each effective insert every revolution

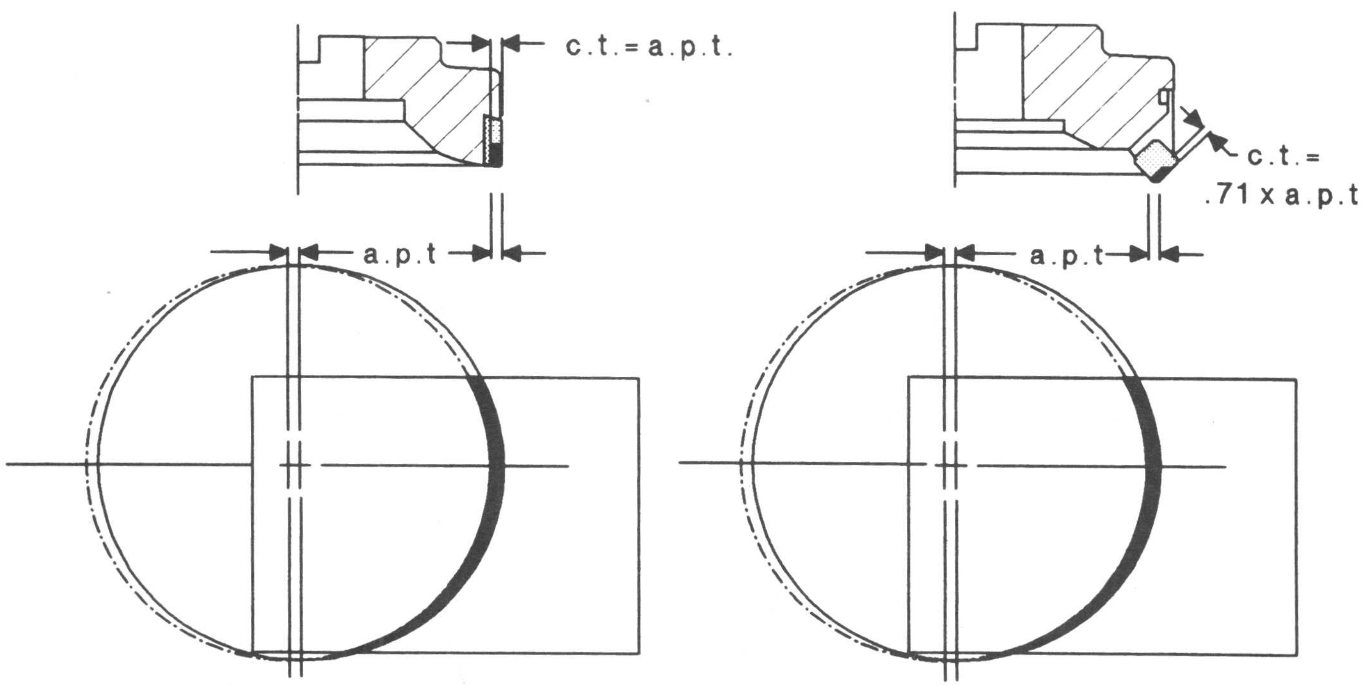
RPM = revolutions per minute

Find	**Given**	**Using**
IPM	a.p.r.,RPM	IPM = a.p.r. x RPM
IPM	RPM, N, a.p.t.	IPM = a.p.t. x N x RPM
a.p.r.	IPM, RPM	a.p.r. = IPM/RPM
RPM	IPM, a.p.r.	RPM = IPM/a.p.r.
RPM	IPM, N, a.p.t.	$\text{RPM} = \frac{\text{IPM}}{\text{N x a.p.t.}}$
N	IPM, RPM, a.p.t.	$\text{N} = \frac{\text{IPM}}{\text{RPM x a.p.t.}}$
a.p.t.	IPM, N, RPM	$\text{a.p.t.} = \frac{\text{IPM}}{\text{RPM x N}}$

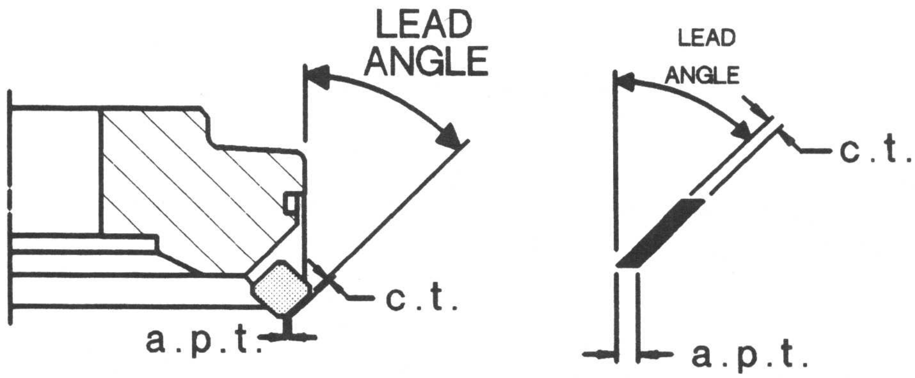
Note : In the formulas shown above IPT can be substituted for a.p.t. and IPR can be substituted in place of a.p.r..

THE LEAD ANGLE EFFECT ON CHIP THICKNESS

Face mills are often available commercially in a variety of lead angles to satisfy part feature requirements and to provide geometry options for different machining situations and conditions. For example, in the high production machining of large cast iron surfaces such as the pan rail, ends or cylinder head joint face of an automotive engine block lead angle cutters of up to 30 degrees are often used to minimize breakout on the exit side of the cut. This is accomplished because the actual chip thickness at the cutter centerline is less than the advance per tooth as shown in the pictures below:

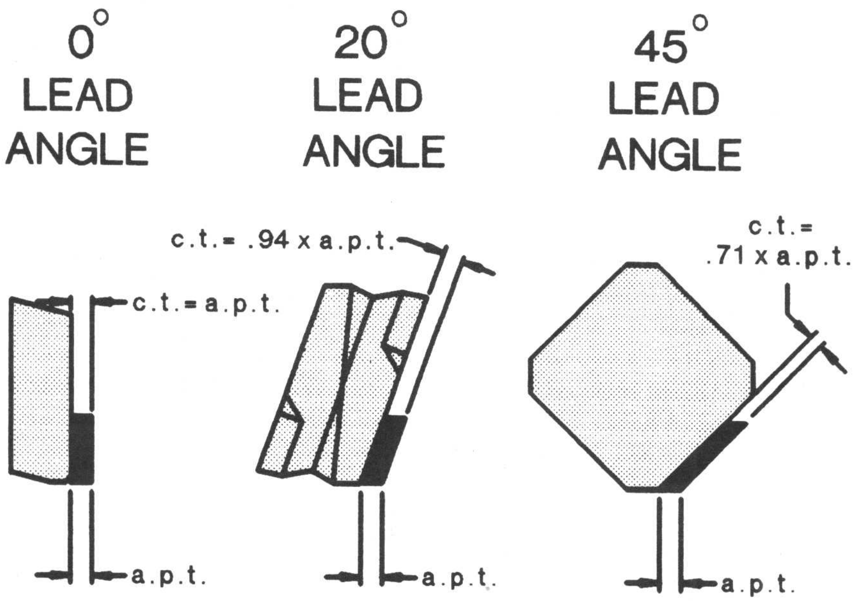


In the left hand picture above, the actual feed per insert or chip thickness is equal to the a.p.t. at the cutter centerline because the cutter has a zero degree lead angle. However, in the right hand picture the actual feed per insert or chip thickness is only 71% of the a.p.t. due to the forty-five degree lead angle on the face mill. The chip thinning effect produced by the lead angle will in many cases allow for an increase in the feed rate (IPM), since the net load on an individual insert is less than the advance per tooth at the cutter centerline. The picture on the following page shows the relationship between the chip thickness (c.t.) and the advance per tooth (a.p.t.):



c.t. = cosine (lead angle) x a.p.t.

The relationships between c.t. and a.p.t. are shown below for some of the more common lead angles now in use:



When catalog recommendations for milling cutter chip thicknesses are used, they are often only considered as a way to establish what the a.p.t. should be without any regard for chip thinning due to a lead angle effect, the following example will show how significantly productivity can be influenced when this effect is overlooked:

<u>EXAMPLE #5</u>

What feed rate should you mill a 300M steel landing gear at with a 4" diameter face mill (containing 8 inserts) if you want to maintain a .006" chip load and the cutter has a 45 degree lead angle?

<u>From example #4:</u>

RPM = 305

a.p.t. = .006

IPM = 14.6

For a 45 degree lead face mill c.t.=.71 x a.p.t.

$$IPM = \frac{c.t.}{.71} \times N \times RPM = \frac{.006}{.71} \times 8 \times 305 = 20.6$$

ANSWER : IPM = 20.6

Note : This represents a 41% increase in feed rate over example #4 where we didn't consider the lead angle and its effect on actual chip thickness.

EXAMPLE #6

In example #5 what feed rate would you use if the lead angle of the cutter was 20 degrees?

From example #5:
RPM = 305
c.t. = .006"
N = 8 inserts

For a 20 degree lead face mill c.t. = .94 x a.p.t.

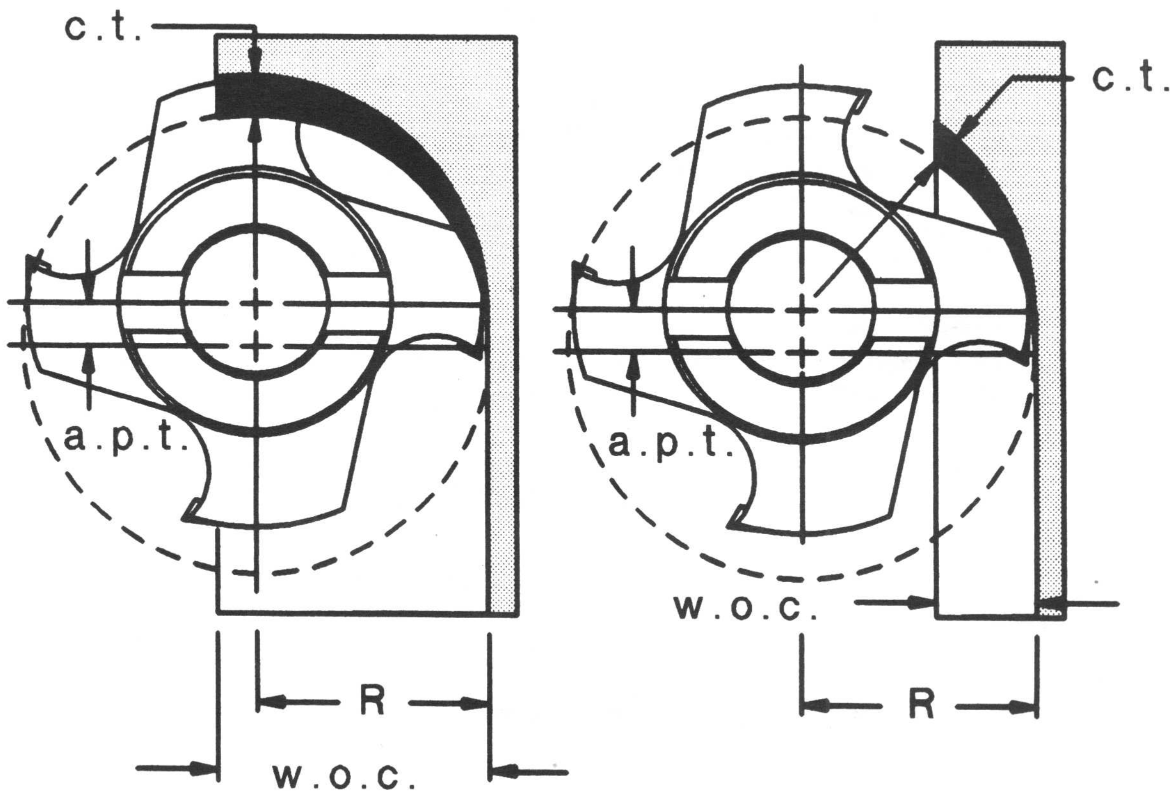
$$IPM = \frac{c.t.}{.94} \times N \times RPM = \frac{.006}{.94} \times 8 \times 305 = 15.6$$

ANSWER : IPM = 15.6

Note : This example demonstrates that the lead angle effect is much smaller with a 20 versus a 45 degree lead angle.

THE WIDTH OF CUT EFFECT ON CHIP THICKNESS

The width of cut in a milling operation can reduce the actual chip thickness an individual insert is subjected to, relative to the a.p.t. at the cutter centerline. This concept is best explained by the following two illustrations:



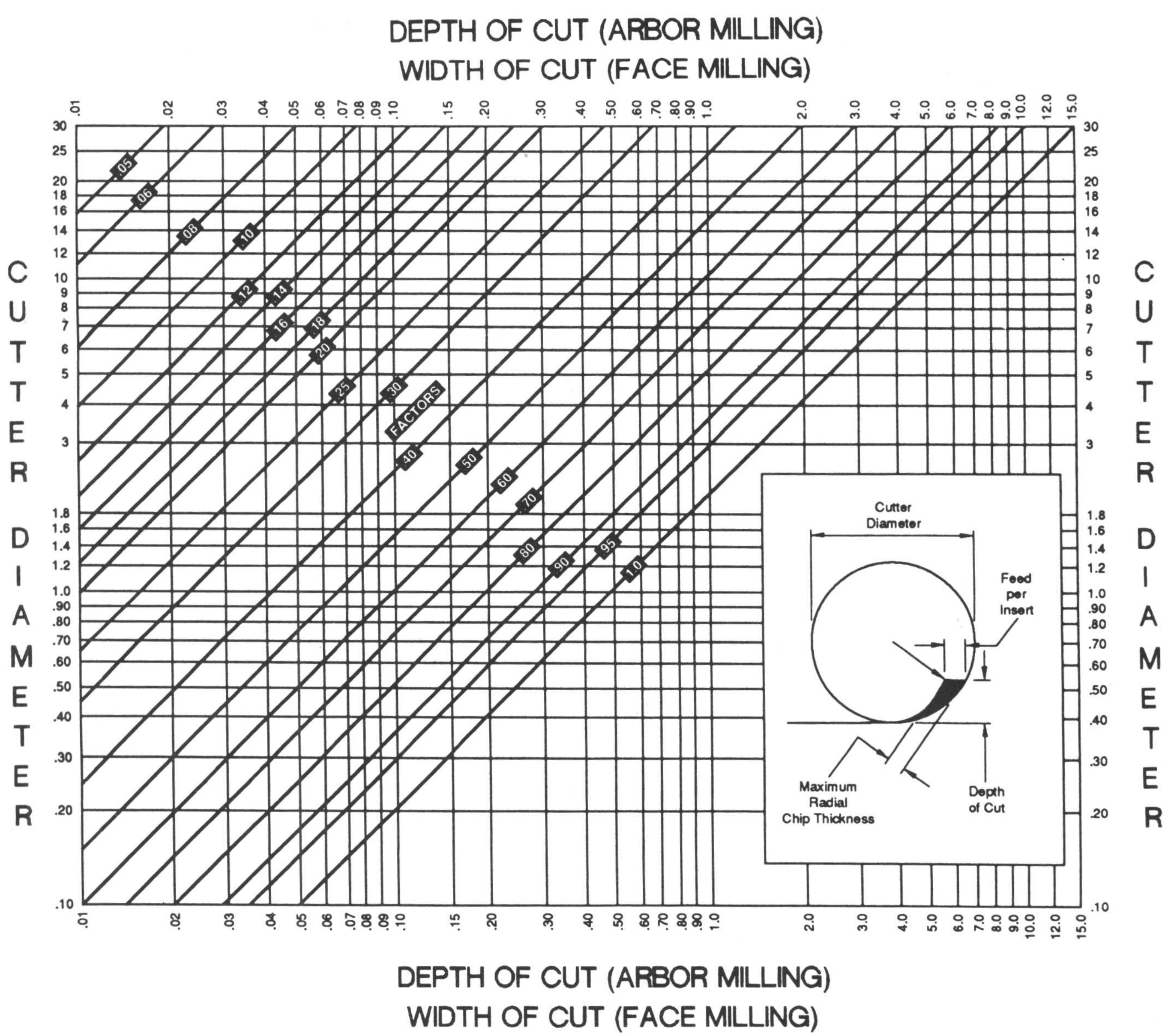
Based on the pictures above the following rules apply:

Rule #1 When the width of cut (w.o.c.) of a milling operation is greater than or equal to the cutter radius, c.t.=a.p.t.

Rule #2 When the width of cut (w.o.c.) of a milling operation is less than the cutter radius, c.t.<a.p.t.

The amount of chip thinning which occurs is dependent on the percentage of the cutter in use in terms of width of cut as stated by rules #1 and #2. The smaller the percentage of the cutter in use the greater the chip thinning effect. This is illustrated by the graph shown below for different cutter diameters and widths of cut (face milling) or radial depths of cut (arbor milling).

MAXIMUM RADIAL CHIP THICKNESS FACTORS



EXAMPLE #7
Use the graph on the previous page to determine the chip thinning factor for a 10" diameter face mill cutting a 2" width of cut?

ANSWER : .8 (c.t.=.8 x a.p.t.)

EXAMPLE #8
If you were milling 6000 series aircraft aluminum at 5089 RPM using a 3" diameter face mill with 3 inserts and you wanted to maintain an .008" chip thickness at a .75" width of cut, what feed rate should you use?

a. If we didn't consider the chip thinning effect, the feed would be:

IPM = a.p.t. x N x RPM = .008" x 3 x 5089 = 120

IPM = 120

b. If we consider the chip thinning effect, than the feed would be:

From the graph c.t. = .85 x a.p.t.

$$IPM = \frac{c.t.}{.85} \times N \times RPM = \frac{.008}{.85} \times 3 \times 5089 = 141$$

ANSWER : IPM = 141

HORSEPOWER REQUIREMENTS IN MILLING

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 which 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:

$$H.P. = Q \times k$$

where:

H.P.= horsepower required at the machine spindle

Q = the metal removal rate in cubic inches per minute

k = the unit power factor in horsepower/cubic inch/minute

The following unit power factors for milling are average values which were established and published by **METCUT RESEARCH:**

MATERIAL	HARDNESS (Bhn or Rc)	UNIT POWER FACTOR ($k = H.P./In.^3/Min.$)
Steels, Wrought & Cast	85-200 Bhn	1.25
Plain Carbon	30-40 Rc	1.70
Alloy Steels	40-50 Rc	2.00
Tool Steels	50-55 Rc	2.90
Cast Irons	110-190 Bhn	.70
	190-320 Bhn	1.25
Stainless Steels, Wrought & Cast	135-275 Bhn	1.55
	30-45 Rc	1.70
Precipitation Hardened Stainless Steel	150-450 Bhn	1.70
Titanium	250-375 Bhn	1.25
High Temperature Alloys (Nickel & Cobalt base)	200-360 Bhn	2.25
Nickel Alloys	80-360 Bhn	2.15
Aircraft Aluminum Alloys	30-150 Bhn	.25

Note : These values are based on milling feeds of .005-.012" a.p.t..

EXAMPLE #9

What feed rate should you select to mill a 2" wide by a .25" depth of cut on aircraft aluminum utilizing all the available horsepower on a 15 H.P. machine using a 3" diameter face mill at 5089 RPM?

$$\mathbf{H.P. = Q \times k}$$

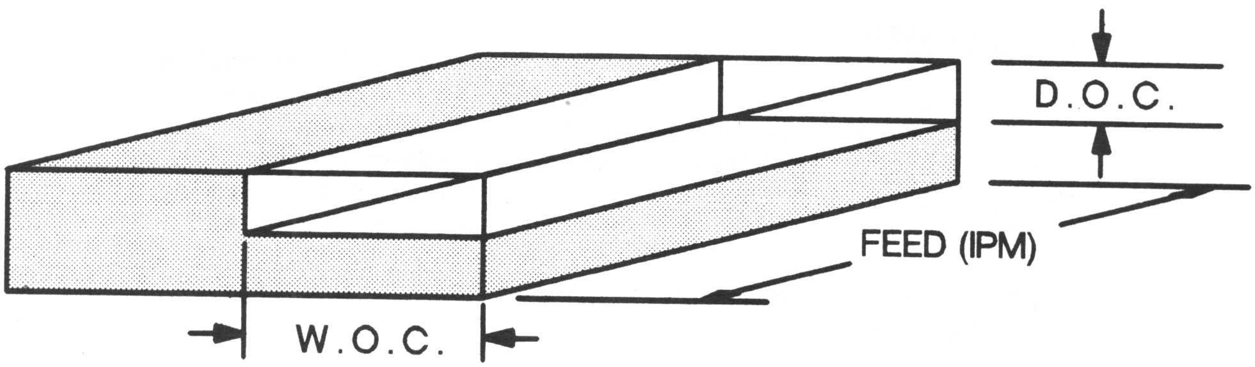
$$k = .25\ H.P./In.^3/Min.\ \text{for aluminum}$$

The maximum possible metal removal rate (Q), for a 15 H.P. machine running an aluminum part is:

$$Q = \frac{H.P.}{k} = \frac{15}{.25} = 60 \text{ In.}^3\text{/Min.}$$

$$Q = 60 \text{ In.}^3\text{/Min.}$$

To remove 60 In.^3/Min., we will need a feed rate of:



$$Q = (D.O.C.) \times (W.O.C.) \times IPM$$

$$IPM = \frac{Q}{(D.O.C.) \times (W.O.C.)} = \frac{60}{.25 \times 2} = 120$$

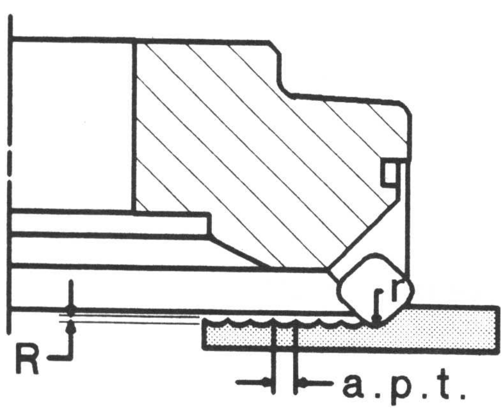
ANSWER : IPM = 120

SURFACE FINISHES IN MILLING

Surface finishes in milling are generally a combination of roughness superimposed along a wave pattern on the workpiece surface. In turning only one insert produces the surface while in milling several inserts overlap as they feed across the workpiece to produce the final surface profile. Since the axial and radial location of all the inserts in an indexable milling cutter are never in perfect alignment, the final surface produced has waves. The portion of the wave produced by the cutter can be exaggerated even further if the machine spindle has runout. The wave produced by the machine spindle is easily observed in the part surface as a repeated mark or pattern dependent on the feed rate of the milling cutter.

RADIUSED INSERTS

The actual shape of an insert cutting edge will be machined into the part surface in both milling and turning operations. Radiused inserts produce grooves in the workpiece surface and, therefore the final finish is dependent on the size of the radius (r) and the feed per insert (a.p.t.), assuming all inserts are exactly in the same plane both radially and axially. Since this type of precise location requires significant setup time and cost, radiused inserts are used on roughing operations.

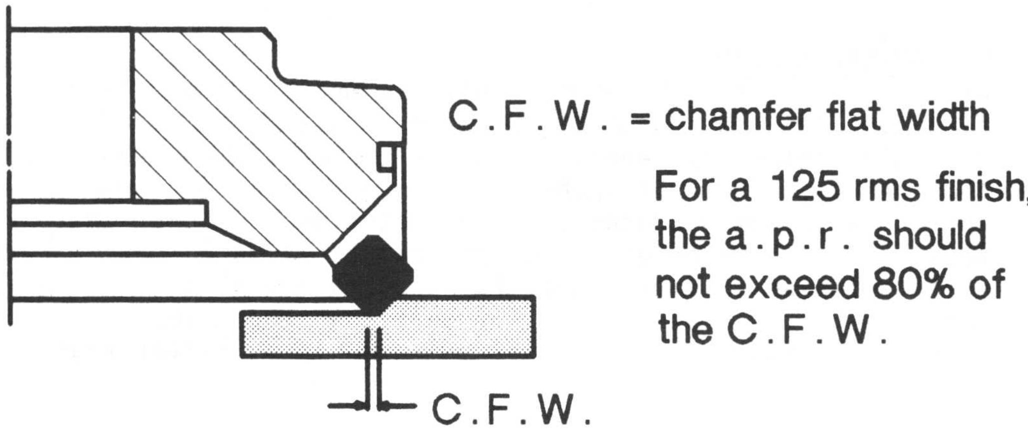


Groove Depth

$$R = 125{,}000 \; \frac{(a.p.t.)^2}{r} \quad (\mu in)$$

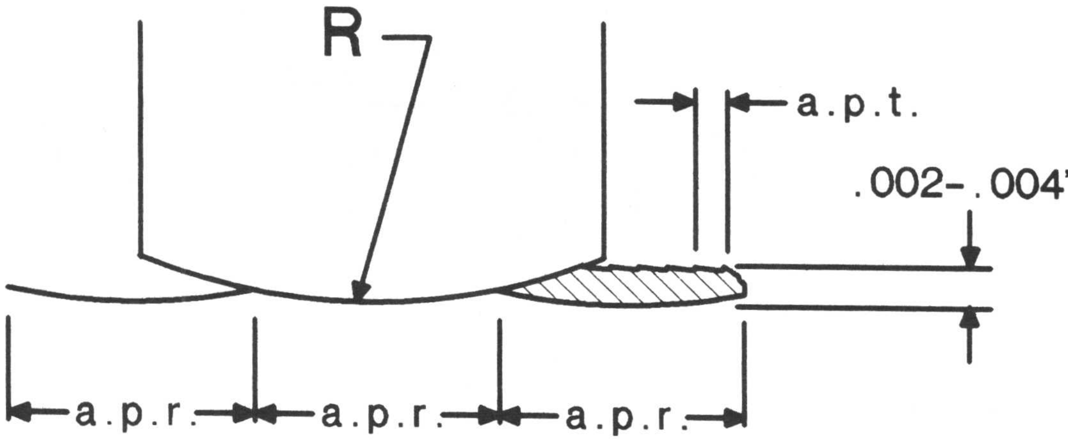
CHAMFERED INSERTS

Inserts with small chamfers or wiping flats are often used when surface finish requirements are a magnitude of 125 micro-inch or less. The chamfer runs parallel to the workpiece surface, and its width must exceed the face mills feed per revolution (a.p.r.). In those cases where the feed per revolution exceeds the width of the chamfered flat, two or more inserts will produce the final surface finish profile and, therefore, it will be totally dependent on the accuracy of their alignment in the axial plane.



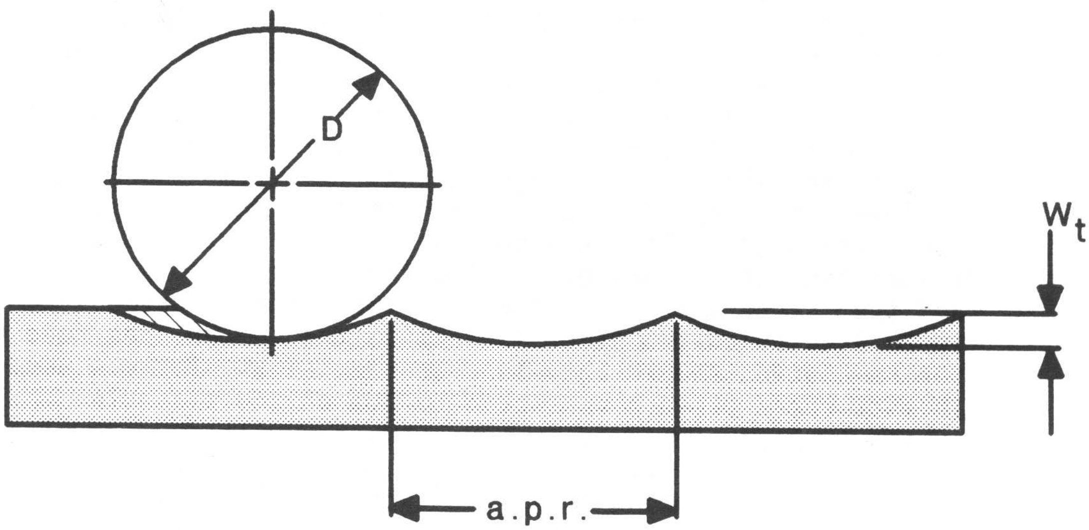
WIPER INSERTS

Wiper inserts are used to obtain fine surface finishes when the feed per revolution greatly exceeds the width of the corner chamfer. In this situation a periphery wiping insert is installed in the cutter body to reduce waviness and minimize the peak to valley height in the surface profile. Wiper inserts are ground with slight crowns (R = 7.8"-15.6") so differences in spindle camber or tilt from application to application will eliminate the typical saw-toothed profile associated with a lack of parallelism between the workpiece and the insert edge. The wiper cutting depth is normally .002-.004" below the other inserts in the cutter. The feed per revolution should be less than the length of the wiper. In most cases the a.p.r. should be 50% of the wiper length. Wipers in large cutter diameters should be aligned axially to insure proper overlap. Cutters using wiper inserts should be used on short chipping materials such as brass and cast iron. The use of wiper blades on steel often causes large axial forces and vibration due to the difficulty in forming a wide chip of only .002-.004" in height.



PERIPHERAL MILLING

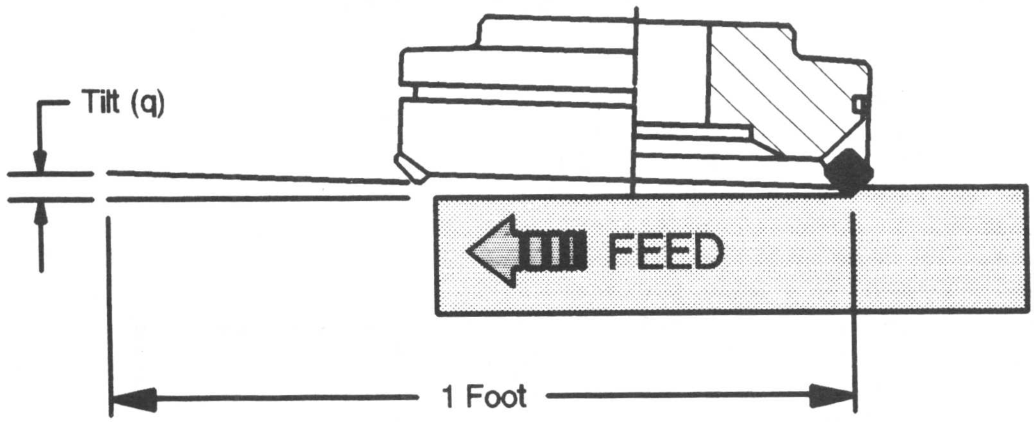
Arbor slotting cutters produce bicurve workpiece profiles on flat surfaces. The depth of the wave produced, using this machining method is dependent on the cutter diameter and the feed per revolution in inches, assuming the tallest insert in the radial plane produces the final machined surface. The equation shown below gives an approximate value of the surface finish in microinches for a surface milled with the periphery of an arbor cutter. If more than one insert acts to produce the final surface, the finish would be better than the calculated value.

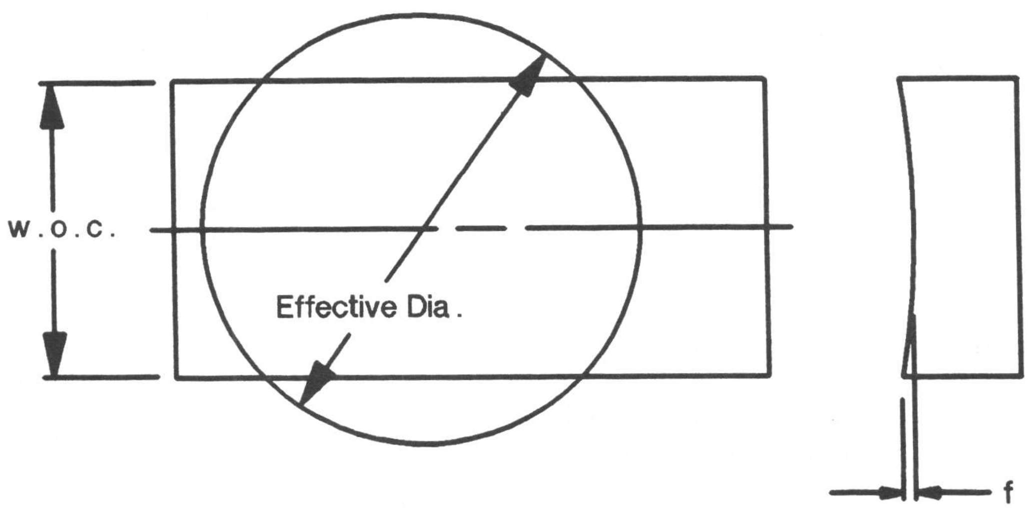


$$W_t = 250{,}000 \frac{a.p.r.^2}{D} \quad (\mu in)$$

SPINDLE TILT

The spindle centerline of most dedicated milling machines is not perpendicular to the machine table. In many cases the spindle is tipped a small amount so the trailing edge of the cutter clears as it is fed across the workpiece. This is referred to as "spindle tilt" and is normally measured in inches/foot. Spindle tilt is used to eliminate "back-cutting or re-cutting" which causes unacceptable surface finishes and reduces tool life due to rubbing. It is important to note that a surface milled on a machine with spindle tilt is not perfectly flat.





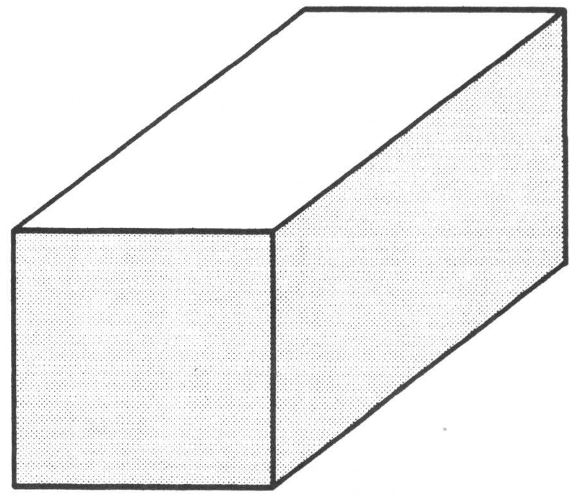
The pictures above show the part dish (f) created by a machine with spindle tilt. This effect (f) is a function of the width of cut, spindle tilt and the effective cutting diameter. Spindle tilt is used mainly on dedicated machines which cut in one direction. Dedicated milling spindles are often found in the automotive industry on cylinder head and block transfer lines and therefore these machines frequently have finish milling spindles with tilt. The amount of spindle tilt required for a given operation vary, however, typical values are .003-.007"/foot.

MILLING OPERATIONS

Milling can be performed on a variety of machines with a vast array of tooling and workpiece configurations. The types of operations performed using milling tools are widespread and they are often dependent on the complexity of the workpiece and the flexibility of the machine tool. For our purposes, we will describe the basic styles of cuts performed with milling tools in order to build a working knowledge of the options available in terms of machining methods, once a part and machine are selected. The pictures shown at the right of each operation description, provide a visual image of an imaginary machined workpiece at the completion of each milling pass.

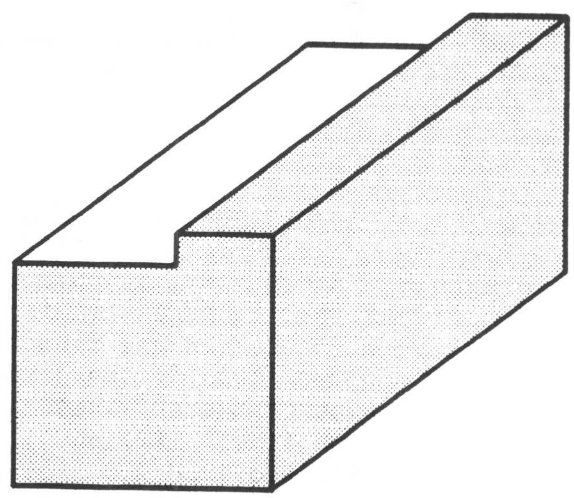
FACE MILLING

This operation is used to mill wide surfaces in a single machine pass perpendicular to the axis of rotation or parallel to the cutter face. The face mill can also be used to machine a wide surface in several overlapping passes, if the cutter diameter is smaller than the finished surface width. In general, when performing a face milling operation the cutter with the largest lead angle and appropriate geometry for the workpiece material should be selected first.



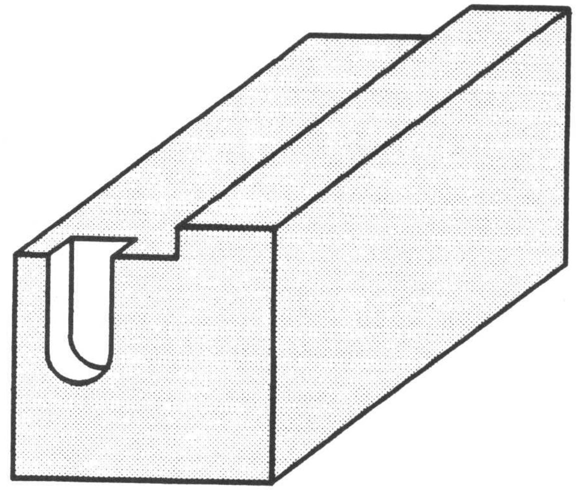
SQUARE SHOULDER MILLING

This operation is the same as face milling, except the workpiece requires a 90 degree shoulder to be cut as a part of a blueprint specification. This type of cut requires the use of a zero degree lead cutter.



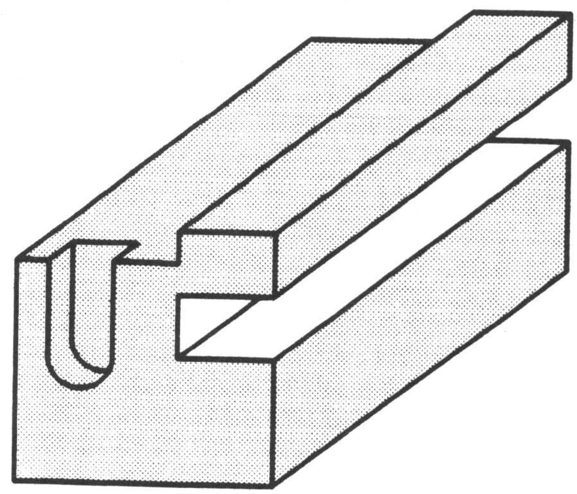
CHANNEL MILLING

This type of milling operation is performed when the entire cutting diameter of an end or face mill is engaged to machine a 90 degree groove in the workpiece. Therefore, this operation involves the use of the end and some or all of the side cutting insert stations on an end mill and all the insert stations on a face mill. In general, when performing this operation with an end mill, the cutting depth each machine pass should never exceed 40% of the end mill diameter.



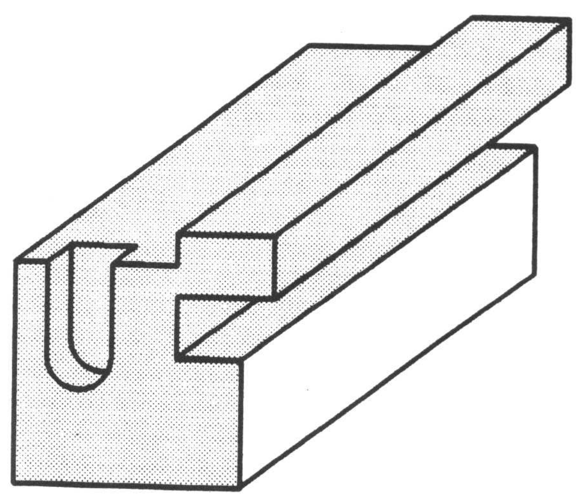
SLOT MILLING

The arbor mill is used to produce slots with its periphery and face mounted inserts. The arbor style mill can produce a variety of slot widths and depths, depending on the workpiece requirements. Although an end mill can perform the same type of operation, an arbor mill is often selected due to its high rigidity and productivity.



SLAB MILLING

This type of milling is performed with the periphery of an end mill, normally around the outer profile of a workpiece. In a slabbing operation, The surface cut by the end mill is parallel to the cutter's axis of rotation. In general, the radial cutting depth of a slab milling application should never exceed 50% of the end mill diameter.



INDEX OF TERMS

Page

REVIEW QUESTIONS

1. Explain the difference between conventional and climb milling.

2. How can you distinguish between a left and right hand milling cutter?

3. What benefit is an insert hone or T-land to a milling operation?

4. Why would you select an arbor mill instead of an end mill to machine a slot?

5. What is the difference between a bevel and lead angle?

6. If a 3" diameter face mill is rotating at 636 RPM, what cutting speed does this represent in SFPM?

7. How does a lead angle effect chip thickness (c.t.) relative to the advance per tooth (a.p.t.) of a face mill?

8. Why is climb preferred over conventional milling for most applications?

9. Under what conditions should conventional milling be used?

10. Does the width of cut (w.o.c.) have any effect on chip thickness in milling operations?

11. What are the differences between NMTBA, CAT & BT tapered adaptors?

12. What percentage of a milling cutter should overhang the workpiece to provide the appropriate entrance angle?

13. Why is spindle tilt used on some machines?

14. What influences horsepower consumption in milling?

15. How should the feed rate of a milling cutter relate to the wiping flat on a milling insert if a fine surface finish is required?

TRAINING MANUAL

WORKPIECE MATERIALS

INTRODUCTION

Metals continue to constitute a majority of the base materials used in the products which dominate our daily lives. They possess many unique properties which over the years have enhanced their value and use by the world's product designers. Some of these properties include strength, toughness, corrosion resistance, hardness and electrical conductivity. Many,or all of these properties might be considered during the material selection stage of a product design process. Therefore, it is essential to comprehend the effects of these or other material properties on machinability (relative ease with which a part can be machined) once a new product design is introduced to the machine shop in the form of a workpiece. In our previous discussions, the workpiece is the piece of metal which has been modified by removing metal chips in one of a vast array of machining methods to arrive at a final part or component shape. The initial piece of metal might be a bar, plate or casting and the final machined component could be an axle shaft, machine table or an automotive cylinder head.

The previous training manuals focused primarily on various cutting tools and machining methods. This manual will examine the effects of specific material properties and conditions on the machining process. In addition, we will compare and index the ease with which different material groups such as cast iron, steels, alloyed steels, aluminum, copper, nickel based alloys, titanium and refractory alloys are machined, along with recommendations in terms of cutting geometry for these groups. Our goal is to present and recommend tool geometries to machine a comprehensive cross-section of metal workpiece, materials which are currently in predominant use in the worlds industrial based economies. In addition, we will review the effects of specific alloying elements on the machinability of a workpiece material, as well as provide reasons for their use in terms of either finished part performance or enhanced manufacturability. This exhaustive overview will encompass workpiece materials which are commonly used in the automotive, aerospace, machine tool, nuclear, mining, heavy equipment and oil field industries. Our sincere desire is to provide a relatively complete reference for questions which ultimately arise regarding a specific workpiece material, or material group.

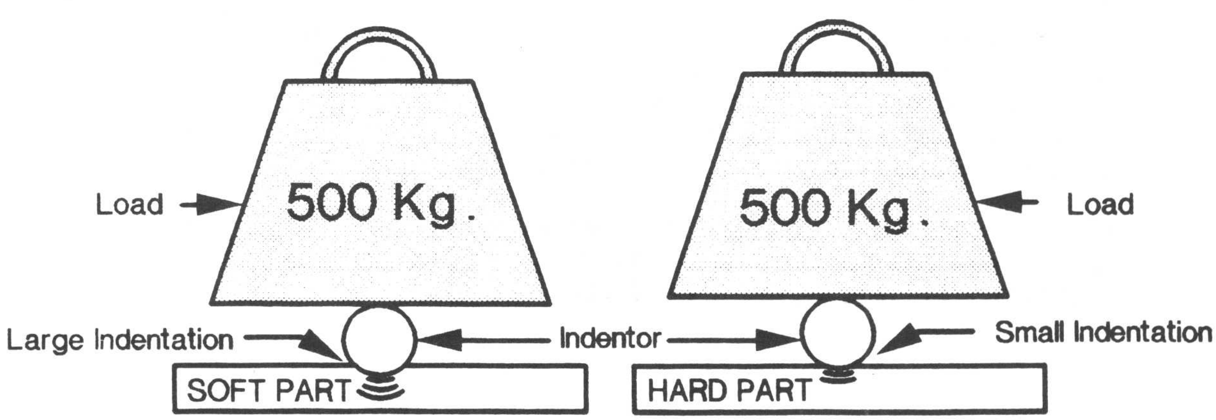
After a thorough review of a material property or group is completed, an explanation of how that subject relates to machining is presented in **bold type.**

CONDITIONS AND PHYSICAL PROPERTIES OF METALS

We will examine those material conditions and properties which have the most significant effect on the relative ease or difficulty associated with the machining of a specific workpiece material. The conditions and properties discussed will often pertain to that group of information often readily available from the raw material supplier to the manufacturing engineer, so an efficient manufacturing plan in terms of cutting speeds and feeds can be established. Material conditions often relate to the hardness, yield strength and tensile strength of the material, which are often a result of heat treatment or the process used to produce the raw material (i.e. casting, rolling, forming or extruding). Physical properties will include those characteristics inherent to the individual material groups, such as the modulus of elasticity, thermal conductivity, thermal expansion and workhardening. Let's review these conditions and properties individually:

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 imbedding a specific size and shaped indentor into the surface of the test material, using a predetermined load or weight. The distance the indentor penetrates the material surface will correspond to a specific Brinell or Rockwell hardness reading. The greater the indentor 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 indentor penetration into the workpiece and thus, by definition, are an indication of an extremely hard part.



The Brinell hardness test involves imbedding a steel ball of a specific diameter, using a kilogram load, in the surface of a test piece. The Brinell (BHN) hardness number 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 comparative 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 indentor sizes and loads. Several different scales exist for the Rockwell method of hardness testing. The three most popular are outlined below in terms of the actual application the test is designed to address:

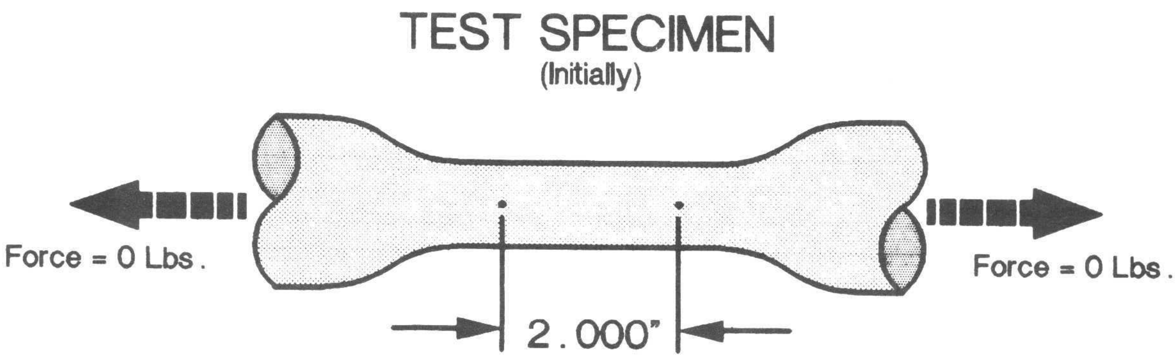
Rockwell Scale	Testing Application
A	For tungsten carbide and other extremely hard materials & thin, hard sheets.
B	For medium hardness low and medium carbon steels in the annealed condition.
C	For materials > than Rockwell "B" 100

(Source : Machinery's Handbook)

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 effected by an increase in workpiece hardness, since the cutting loads and temperatures rise for a specific cutting speed with part hardness, thereby reducing edge life. In drilling and turning, the added cutting temperature is detrimental to tool life, since it produces excess heat causing accelerated edge wear, while 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 when the load is removed and the marks will not return to their initial distance apart. At this point, permanent set or deformation of the test specimen has taken place.



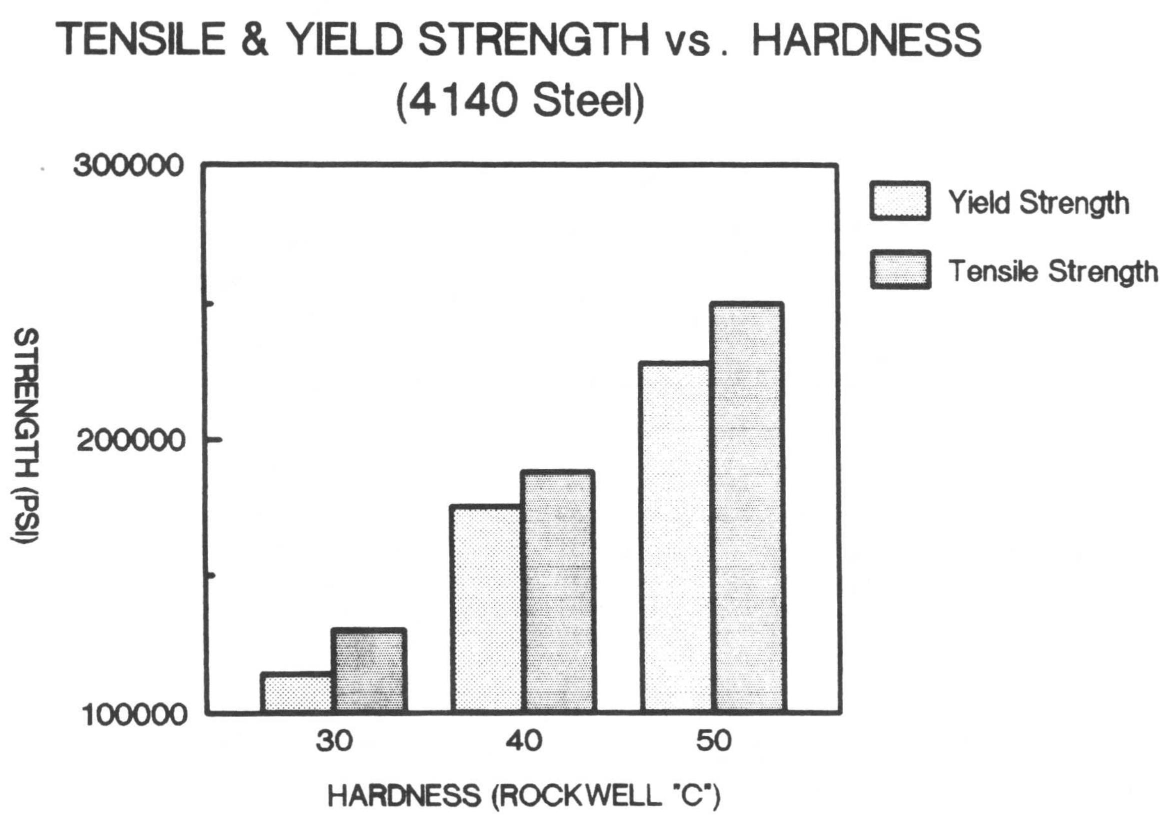
Yield strength is measured just prior to the point before permanent deformation takes place. Yield strength is stated in pounds per square inch (psi) and is determined by dividing the load just prior to permanent deformation by the cross-sectional area of the test specimen. This material property has 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 a 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 materials 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 as well as 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.

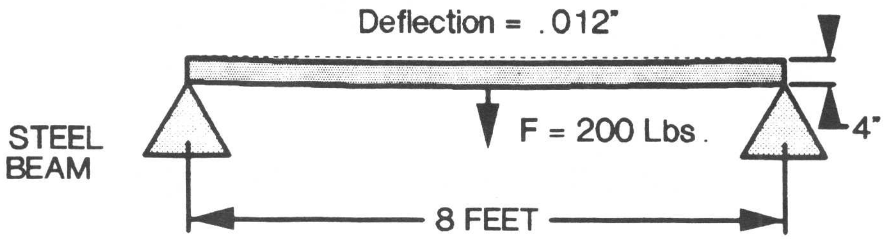
The most conclusive evidence that hardness, yield and tensile strength are physical conditions of materials is illustrated by the following graph of a heat treated 4140 steel at three hardness levels:

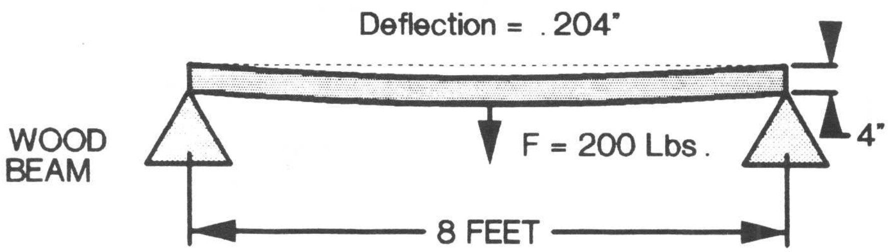


Just as increased yield strength implied higher cutting forces during machining operations, the same could be said for increased tensile strength. Again, as workpiece tensile strength is elevated, stronger cutting edge geometries are required for productive machining and acceptable tool life.

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 and 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 lb. weight hanging in the middle will sag 17 times more than a beam made with the same dimensions 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. For further details on this material property see page 38, of the **"~~HERTEL~~ TRAINING MANUAL FOR TURNING"**. The modulus of elasticity for some common materials is shown below:

THE MODULUS OF ELASTICITY FOR SOME COMMON MATERIALS

Material	Modulus in Psi
Wood (Pine)	1,760,000
Aluminum (99.9+% Pure)	10,000,000
Cast Iron (Gray)	13,000,000
Titanium (99.5+% Pure)	14,900,000
Copper (99.9+% Pure)	16,000,000
Steel (1020)	30,000,000
Mallory (Heavy Metal)	45,000,000
Carbide (C2)	90,000,000

General manufacturing practice dictates that productive machining of a workpiece material with a relatively moderate modulus of elasticity normally requires positive or highly positive rake cutting geometries. Positive cutting geometries produce lower cutting forces and, therefore, chip formation is enhanced on elastic materials 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. This phenomenon is amplified when the workpiece material has a relatively moderate or small modulus of elasticity since, by definition, the modulus of elasticity of a material is directly related to the rate at which it moves or deflects under load.

THERMAL CONDUCTIVITY

Materials are frequently labeled as being either heat conductors or insulators. Conductors tend to transfer heat from a hot to 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. The thermal conductivity for some common materials is shown below:

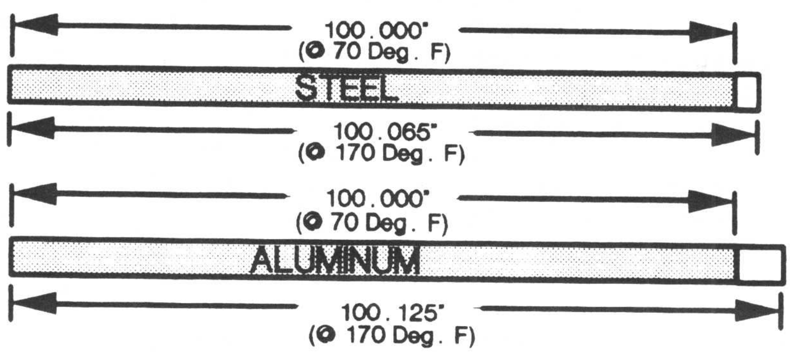
THE THERMAL CONDUCTIVITY FOR SOME COMMON MATERIALS

Material	Thermal Conductivity (BTU/Hr-Ft-°F)
Copper (99+% Pure)	223.0
Aluminum (99.9+% Pure)	132.0
Cast Iron	30.0
Steel (1020)	28.8
Stainless Steel 304	9.4
Inconel 718	6.5
Inconel 625	5.7
Titanium 6Al-4V	3.8
Celotex Insulation	.028

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 copious 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 a metal expands can be determined using the material's thermal 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 deg. 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. See the drawings below:



The thermal expansion coefficients for some common materials are shown below:

THE THERMAL EXPANSION COEFFICIENT FOR SOME COMMON MATERIALS

Material	Thermal Expansion Coefficient (Inch/Inch Deg. F)
Aluminum (99.9+% Pure)	.0000125
Steel (1020)	.0000065
Cast Iron	.0000056
Titanium 6Al-4V	.0000048

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.

WORKHARDENING

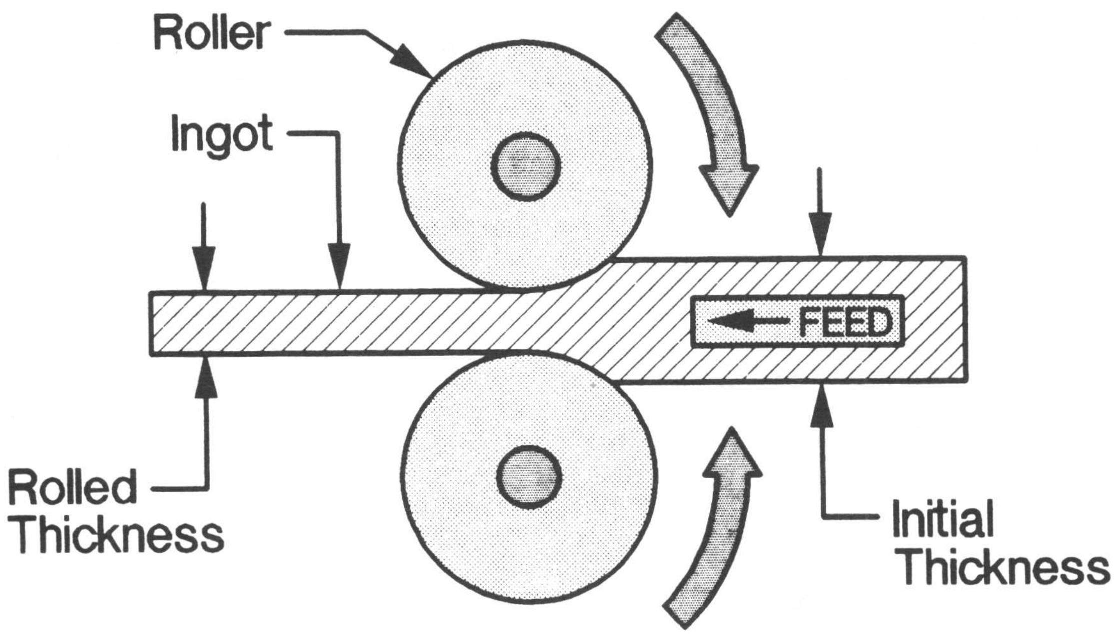
Many metals exhibit a physical characteristic which produces dramatic increases in hardness due to cold work. Cold work involves changing the shape of a metal object by bending, shaping, rolling or forming. As the metal is shaped, internal stresses develop which act to harden the part. The rate and magnitude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

The machining of workpiece materials with workhardening 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 workhardening 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.

WROUGHT AND CAST MATERIALS

The term "wrought" refers to the hammering or forming of materials into premanufactured 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. The processing of these parts involves rolling, drawing or extrusion. In addition, forged parts are considered wrought materials since they are normally hammered into a premachined configuration.

A two inch thick four foot wide by eight foot long wrought aluminum plate begins as an aluminum ingot (casting) which is up to 20 feet in length, eight inches thick and eight feet wide. The finished plate dimensions are achieved by reducing the thickness of the ingot using huge rolling mills which utilize pressure and rotational motion to feed the ingot back and forth through the mill while systematically squeezing the aluminum into a thinner and longer plate. See the diagram below:



As the length of the plate becomes unmanageable, the excess material is trimmed with an enormous shear (knife).

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. Nearly all metal alloys are available in some type of cast form, although the techniques used to produce cast parts from different alloys will vary significantly.

MACHINABILITY

The term "machinability" is a relative measure of how easily a material can be machined when compared to 160 Brinell A.I.S.I. B1112 free machining low carbon steel. The American Iron and Steel Institute 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 machinability rating. The chart below shows a variety of materials and their specific machinability ratings:

Material	Hardness	Machinability Rating
6061-T Alum.	-	190%
7075-T Alum.	-	120%
B1112 Steel	160 BHN	100%
416 Stainless Stl.	200 BHN	90%
1120 Steel	160 BHN	80%
4140 Steel (leaded)	187 BHN	70%
1020 Steel	148 BHN	65%
8620 Steel	194 BHN	60%
304 Stainless Stl.	160 BHN	40%
17-4 PH Stainless Stl.	388 BHN	28%
Inconel X	360 BHN	15%
Rene 41	215 BHN	15%
Waspalloy	270 BHN	12%
A-286	300 BHN	10%
Hastelloy X	197 BHN	9%

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 iron 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.

Cast iron produces short discontinuous chips, powder or flakes when machined. The chemical makeup of cast iron varies widely with the cooling rate of the part. Hardness and grain structure within a single casting can also change dramatically based on the symmetry or uniformity in cross-sectional thickness of the cast part. Since many complex castings have a diverse mixture of thick and thin cross-sections, the cooling rate of these areas will vary widely and thus, so will their final hardness and grain structure. Alloying elements, cooling rates, carbon content (in the form of graphite or cementite) and silicon content will have a dramatic effect on the physical properties of cast iron. We will discuss how these different combinations of manufacturing and alloying procedures influence the final cast iron grade.

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. These materials contain from 1.7-4.5 percent carbon and 1-3 percent silicon.

Gray cast irons are grouped and then categorized by strength according to the American National Standard Institutes (ANSI) and the American Society for Testing and Materials (ASTM) standard A48-76. These 2 groups are as follows:

GRAY CAST IRON

Class No.	Minimum Tensile Strength
20A	20,000 psi
20B	" "
25A	25,000 psi
25B	" "
30A	30,000 psi
30B	" "
35A	35,000 psi
35B	" "

Based on the class numbers shown, the first two digits in the class number indicate the minimum tensile strength of the material in (1000s) of psi. This first group of gray cast iron class numbers represents a set of materials which exhibit good machinability, a relatively low modulus of elasticity, and excellent dampening properties due to their graphite contents. The second group of gray cast iron materials have lower machinability ratings, a higher modulus of elasticity and lower dampening capacity. In general they are more difficult to machine based on their higher strength levels. The second group is shown below:

GRAY CAST IRON

Class No.	Minimum Tensile Strength
40B	40,000 psi
40C	" "
45B	45,000 psi
45C	" "
50B	50,000 psi
50C	" "
60B	60,000 psi
60C	" "

Again, the tensile strength of these materials is indicated by the first two digits of the grade classification in 1000's of psi. Both groups of gray cast irons are characterized by high contents of carbon which is dispersed throughout the material in the form of graphite. Graphite flake formation is dependent on the cooling rate of the casting.

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. This material has very high compressive strength and is able to withstand 200,000 psi loads, while gray iron castings have lower compressive strength levels of 65,000-160,000 psi. 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. The terms graphitization or malleablizing are also frequently used to describe this transformation process. Hence, the name Malleable Iron. 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 broad scope of applications. The following list of strength levels for malleable iron castings is specified by ANSI/ASTM Standard A47-77:

MALLEABLE CAST IRON

Casting Grade No.	Minimum Yield Strength	Minimum Tensile Strength
32520	32,500 psi	50,000 psi
35018	35,000 psi	53,000 psi

Pearlitic malleable iron can be produced by prematurely ending the annealing of white iron castings, or by reheating an existing malleable iron casting to a high temperature where graphite particles transform into pearlite and therefore, several forms of carbon are left to exist within a single casting. Pearlitic casting specifications are covered under ASTM Specification A 220-79 and are as follows:

MALLEABLE CAST IRON

Casting Grade No.	Minimum Yield Strength	Minimum Tensile Strength
40010	40,000 psi	60,000 psi
45008	45,000 psi	65,000 psi
45006	45,000 psi	65,000 psi
50005	50,000 psi	70,000 psi
60004	60,000 psi	80,000 psi
70003	70,000 psi	85,000 psi
80002	80,000 psi	95,000 psi
90001	90,000 psi	105,000 psi

The last two numbers in the casting grade number are a measure of how ductile (ability to stretch or elongate without breaking) these materials are in terms of a percentage. For example, grade 40010 has a 10% level of ductility and grade 90001 has a 1% level. By definition, a grade 40010 malleable iron casting will stretch without breaking to a much greater extent than one made of grade 90001.

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. Malleable casting hardness for non-pearlitic castings is often 110-160 Brinell, while pearlitic castings are normally 160-320 Brinell. Tool life and cutting speeds should be greater on non-pearlitic malleable iron castings than on their pearlitic counterparts, based on the aforementioned differential in hardness.

NODULAR CAST IRON

Nodular or "ductile" iron is used to manufacture a wide range of automotive engine components including cam shafts, crank shafts, bearing caps and cylinder heads. This material 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. The properties of this material vary significantly from those of gray cast iron since the graphite present in this material is in a ball shape instead of flakes. Since this material exhibits the toughness of many low carbon steels, it is often an economical cast material alternative to a fully machined steel part.

Ductile iron is available in several physical forms in terms of hardness and strength based on the materials heat treatment. In addition, since this material exhibits many of the desirable casting (castability, or ease of flow after it is heated to melting point) and machining qualities of gray cast iron, it is a popular material choice for components which are subjected to intermittent shock loads.

Ductile cast irons are covered by ASTM Specification A536-80 and some of the resulting grades and physical properties are as follows:

Nodular Cast Iron

Casting Grade No.	Minimum Yield Strength	Minimum Tensile Strength
60-40-18	40,000 psi	60,000 psi
65-45-12	45,000 psi	65,000 psi
80-55-06	55,000 psi	80,000 psi
100-70-03	70,000 psi	100,000 psi
120-90-02	90,000 psi	120,000 psi

The ductility of these castings is detailed in the last two digits of the casting grade classification, just as it was for malleable iron. Again, by definition, a grade 60-40-18 nodular iron casting will stretch without breaking to a much greater extent than one made of grade 120-90-02.

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. A 60-40-18 nodular iron casting will have a typical hardness of 140-190 Brinell, while a 120-90-02 grade casting can attain a hardness level of 330-400 Brinell. The manufacturability or machinability of a nodular iron casting is totally dependent on the iron grade. 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.

STEEL

Steel materials are comprised mainly of iron and carbon, often with a modest mixture of alloying elements. The design engineer often selects steel as a workpiece material due to its relatively low cost and versatility in terms of physical properties following heat treatment, or as a result of alloying (adding elements to the iron-carbon base material to produce improvements in physical properties). 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% carbon, to 4.5% carbon. Steel has a typical carbon content .05-1.5%.

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 STEELS

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.

Low carbon steels of the plain carbon type contain .06-.27% carbon. The lower carbon steels in this category include AISI or SAE (Society of Automotive Engineers) **1006, 1008, 1010** and **1015.** These materials, due to their high ductility, are often selected for applications where forming or drawing is required, such as automobile body panels, covers for transmissions, rivets, cold headed products and wire.

The remaining steels in this category include AISI or SAE **1016, 1017, 1018, 1019, 1020, 1021, 1022, 1023, 1024, 1025, 1026** and **1027.** These steels have reduced levels of ductility when compared to the lower carbon steels (.06-.015% carbon) in this classification. Therefore, these materials are used to make components where the higher levels of carbon produce the added strength necessary for the application. The steels in this category are often carburized (hardened on the outer surface for added strength and durability) for more demanding operations. SAE 1020 and 1025 steel is often used for lower strength fasteners such as bolts.

Medium carbon steels of the plain carbon type contain .30-.52% carbon. These materials would include, AISI or SAE **1030, 1033, 1034, 1035, 1036, 1038, 1039, 1040, 1041, 1042 1043, 1045, 1046, 1049, 1050** and **1052**. The increased carbon content of this group of steels makes them an excellent choice for forged and cold formed products where they can be heat treated to a moderate level and the desired hardness of the finished component is obtained by cold working the part. These steels are commonly used in spring wire, crankshafts, drive keys, cold headed rods and ring gears.

High carbon steels of the plain carbon type, contain .55-.95% carbon. These materials would include, AISI or SAE **1055, 1060, 1062, 1064, 1065, 1066, 1070, 1074, 1078, 1080, 1085, 1086, 1090** and **1095**. These steels are almost never used for cold forming processes due to their relatively high carbon contents. However, they are often machined following a preliminary heat treating process. Common parts manufactured from these steels include gears, snap rings, shafts, clutch disks, and axles.

The machinability of plain carbon steels is primarily dependent on the carbon content of the material and its heat treatment. Those materials in the low carbon category are extremely ductile, which creates tremendous problems in chip breaking on turning and drilling operations. In some cases, it is impossible to attain acceptable chip control on these materials in a non-heat treated condition. This often occurs when the chip begins welding small deposits of steel to the cutting edge (built up edge). 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. When limited horsepower is available, double positive milling geometries can be used successfully on non-heat treated low and medium carbon plain carbon steels. In turning and drilling operations on these materials, negative or neutral geometries should be used whenever possible. The added edge strength available from these types of tools is often required to withstand the increased chip thickness necessary on low and medium carbon plain steels to produce acceptable chip control. High carbon steels of the plain carbon type can be successfully machined with positive, neutral or negative geometries in a non-heat treated condition. When these materials are heat treated to 35 Rockwell "C" or above, negative or positive/negative geometries should be used whenever available. 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 a 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. This group of alloyed steels is designated SAE or AISI 23xx or 25xx. These alloys are often used to manufacture hardened gears, axle shafts and hardened studs in SAE No. 2330, 2340 and 2345.

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. Chromium steels typically have a fine grain structure with exceptional wear resistance and are covered by SAE or AISI 50xx, 51xx, 50xxx, 51xxx, and 52xxx. SAE 52100 is a popular material used in the manufacture of ball bearing balls, bearing races and bearing rollers, while SAE 5150 is frequently used on oil-hardened gears and shafts.

Manganese - This category of alloyed steels possesses a greater strength level than nickel alloyed steels and improved toughness when compared to chromium alloyed steels. Manganese alloyed steels are covered under SAE or AISI No. 13xx. SAE 1320 is used in the production of spline shafts.

Vanadium - Vanadium alloyed steels are stronger, harder and tougher than their manganese counterparts. This group of materials, however, surrenders 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 benefits 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. The SAE or AISI designations of 40xx and 44xx cover molybdenum alloy steels. SAE 4042 is often used to manufacture cold headed products such as bolts while coil springs, leaf springs and axle shafts are made from SAE 4063.

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. The chrome-nickel alloyed steels are covered by SAE or AISI designations 31xx, 32xx, 33xx and 34xx. SAE 3135, 3140 and 3141 are popular steels used in the manufacture of steel axle shafts and crankshafts.

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 -Vanadium alloy steels are categorized as SAE or AISI 61xx. SAE 6150 alloyed steel is used in the manufacture of leaf springs, shafts and heavy duty gearing.

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. SAE or AISI No. 41xx designates chrome-molybdenum alloy steels. SAE 4140 is a very popular steel used on a wide variety of shafts such as axle, transmission, or propeller.

Nickel-Molybdenum - The properties of this material are similar to chrome-molybdenum alloyed steels except for one, its increased toughness. Nickel-Molybdenum steels fall under the SAE or AISI designation of 46xx or 48xx. SAE 4615 and 4620 are also used on heavy duty and cam shafts. SAE 4810 and 4815 is frequently used on heavy duty bolts and chain pins for transmissions.

The following chart provides all of the SAE and AISI steel designations :

SAE & AISI STEEL DESIGNATIONS

SAE or AISI NO.	MATERIAL COMPOSITION
	CARBON STEELS
10xx	Plain Carbon (Mn 1% max.)
11xx	Free machining plain carbon -resulfurized
12xx	Resulfurized & Rephosphorized
15xx	Plain carbon (Maximum Mn 1-1.65%)
	MANGANESE STEELS
13xx	Mn 1.75%
	NICKEL STEELS
23xx	Ni 3.50%
25xx	Ni 5.00%
	NICKEL-CHROMIUM STEELS
31xx	Ni 1.25% ; Cr 0.65% & 0.80%
32xx	Ni 1.75% ; Cr 1.07%
33xx	Ni 3.50% ; Cr 1.50% & 1.57%
34xx	Ni 3.00% ; Cr 0.77%
	MOLYBDENUM STEELS
40xx	Mo 0.20% & 0.25%
44xx	Mo 0.40% & 0.52%
	CHROMIUM-MOLYBDENUM STEELS
41xx	Cr 0.50%,0.80%,0.95%;Mo .12%,0.20%,0.25%,.30%
	NICKEL-CHROME MOLYBDENUM STEELS
43xx	Ni 1.82%; Cr 0.50% & 0.80%; Mo 0.25%
47xx	Ni 1.05%; Cr 0.45% ; Mo 0.20% & 0.35%
86xx	Ni 0.55%; Cr 0.50% ; Mo 0.20%
87xx	Ni 0.55%; Cr 0.50% ; Mo 0.25%
88xx	Ni 0.55%; Cr 0.50% ; Mo 0.35%
	NICKEL-MOLYBDENUM STEELS
46xx	Ni 0.85% & 1.82% ; Mo 0.20% & 0.25%
48xx	Ni 3.50% ; Mo 0.25%
	CHROMIUM STEELS
50xx	Cr 0.27%, 0.40%, 0.50% & 0.65%
51xx	Cr 0.80%, 0.87%, 0.92%, 0.95%, 1.00% & 1.05%
50xxx	Cr 0.50%; C 1.00% minimum
51xxx	Cr 1.02%; C 1.00% minimum
52xxx	Cr 1.45%; C 1.00% minimum
	CHROMIUM-VANADIUM STEELS
61xx	Cr 0.60%, 0.80% & 0.95%; V 0.10% & 0.15% min.
	TUNGSTEN-CHROMIUM ALLOYS
72xx	W 1.75%; Cr 0.75%
	SILICON MANGANESE STEELS
92xx	Si 1.40% & 2.00%; Mn 0.65%, 0.82% & 0.85%
	LEADED STEELS
xxLxx	L indicates a leaded steel

C=Carbon
Cr=Chromium
Mn=Manganese
Mo=Molybdenum
Ni=Nickel
Si=Silicon
V=Vanadium
W=Tungsten

(Source : Machinery's Handbook)

The steel designations on the previous page follow the standard four digit nomenclature used by both the Society of Automotive Engineers (SAE) and the American Iron and Steel Institute. The first two digits in each number provide an indication of the chemical composition of the material, while the last two digits are often an indication of the carbon content in hundredths of a percent. For example 6150 steel is a chrome-vanadium steel with a carbon content of .50%. The five digit codes are used for special alloy steels, while those materials containing an "L" have lead added to enhance their machinability.

The machinability of alloy steels varies widely, depending on their hardness and chemical compositions. Normally, fine grained materials tend to be tougher and thus more difficult to machine. However, Chromium, Molybdenum and Chrome-Molybdenum alloy steels are exceptions to this rule and, therefore, they tend to be more machinable than some of the other alloyed steels. Vanadium, Nickel, Chrome-Nickel, Chrome-Vanadium and Nickel-Molybdenum steels have relatively high toughness, which inhibits their machinability. 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. Alloyed steels with hardness levels exceeding 40 Rockwell "C" should be milled with either double negative geometries, or positive (radial) and negative (axial) geometries with heavily T-landed insert edge preparations. In turning operations, double negative or neutral geometries should be used on softer alloy steels, and when hardness (to 40 Rockwell "C") is increased, heavily chamfered edges are often required. 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. The following chart details the most popular tool steels available along with their letter designations:

SAE & AISI TOOL STEEL DESIGNATIONS

Category Designation	Letter Symbol	Group Designation
High Speed Tool Steels	M	Molybdenum Types
	T	Tungsten Types
Hot Work Tool Steels	H1-H19	Chromium Types
	H20-H39	Tungsten Types
	H40-H59	Molybdenum Types
Cold Work Tool Steels	D	High Carbon, high chromium
	A	Medium Alloy, air hardening
	O	Oil hardening type
Shock Resisting Tool Steels	S	....
Mold Steels	P	....
Special Purpose Tool Steels	L	Low alloy types
	F	Carbon Tungsten types
Water Hardening Tool Steels	W	....

(Source : Machinery's Handbook)

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. Low shear positive cutting geometries and moderate chip loads should be considered on D, A and O series cold work tool steels to avoid the disastrous effects of prehardening or burnishing of the material during the machining process due to excessive heat or rubbing.

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. We will evaluate each of these categories individually.

Austenitic Stainless Steels contain chrome, nickel and manganese as their primary alloying elements. These materials are hardened by cold working, and softened by heat treating. This group of materials falls under the AISI 200 & 300 series numerical code and these groups are known for their tremendous ductility. Their chemical compositions are as follows:

CHEMICAL COMPOSITION OF AUSTENITIC STAINLESS STEELS

	Chemical Analysis % (Maximum unless noted otherwise)							
AISI#	C	Mn	P	S	Si	Cr	Ni	Mo	Other
201	.15	5.5/ 7.5	.06	.03	.75	16.0/ 18.0	3.5/ 5.5		.25N
202	.15	7.5/ 10.0	.06	.03	.75	17.0/ 18.0	4.0/ 6.0		.25N
205	.12/ .15	14/ 15.5	.06	.03	.75	16.5/ 18.0	1.0/ 1.75		.32/ .40N
301	.15	2.0	.045	.03	.75	16.0/ 18.0	6.0/ 8.0		
302	.15	2.0	.045	.03	.75	17.0/ 19.0	8.0/ 10.0		.10N
302B	.15	2.0	.045	.03	2.0/ 3.0	17.0/ 19.0	8.0/ 10.0		
303	.15	2.0	.20	.03	.75	17.0/ 19.0	8.0/ 10.0		
303Se	.15	2.0	.20	.06	1.0	17.0/ 19.0	8.0/ 10.0		.15 (min)Se
304	.08	2.0	.045	.03	.75	18.0/ 20.0	8.0/ 10.5		.10N

CHEMICAL COMPOSITION OF AUSTENITIC STAINLESS STEELS CONTINUED

	Chemical Analysis % (Maximum unless noted otherwise)							
AISI#	C	Mn	P	S	Si	Cr	Ni	Mo	Other
304L	.03	2.0	.045	.03	.75	18.0/ 20.0	8.0/ 12.0		.010N
304Cu	.08	2.0	.045	.03	.75	17.0/ 19.0	8.0/ 10.0		3.0/ 4.0 Cu
304N	.08	2.0	.045	.03	.75	18.0/ 20.0	8.0/ 10.5		.010/ .16N
305	.12	2.0	.045	.03	.75	17.0/ 19.0	10.5/ 13.0		
308	.08	2.0	.045	.03	1.0	19.0/ 21.0	10.0/ 12.0		
309	.20	2.0	.045	.03	1.0	22.0/ 24.0	12.0/ 15.0		
309S	.08	2.0	.045	.03	1.0	22.0/ 24.0	12.0/ 15.0		
310	.25	2.0	.045	.03	1.5	24.0/ 26.0	19.0/ 22.0		
310S	.08	2.0	.045	.03	1.5	24.0/ 26.0	19.0/ 22.0		
316	.08	2.0	.045	.03	.75	16.0/ 18.0	10.0/ 14.0	2.0/ 3.0	.10N
316L	.03	2.0	.045	.03	1.5	16.0/ 18.0	10.0/ 14.0	2.0/ 3.0	.10N
316F	.08	2.0	.200	.10	1.0	16.0/ 18.0	10.0/ 14.0	1.75/ 2.50	
316N	.03	2.0	.045	.03	.75	16.0/ 18.0	10.0/ 14.0	2.0/ 3.0	.10/ .16 N
317	.08	2.0	.045	.03	.75	18.0/ 20.0	11.0/ 15.0	3.0/ 4.0	.10N
317L	.03	2.0	.045	.03	.75	18.0/ 20.0	11.0/ 15.0	3.0/ 4.0	.10N
321	.08	2.0	.045	.03	.75	17.0/ 19.0	9.0/ 12.0		5(C+N)min/ .7max(Ti).1N

CHEMICAL COMPOSITION OF AUSTENITIC STAINLESS STEELS CONTINUED

	Chemical Analysis % (Maximum unless noted otherwise)								
AISI#	C	Mn	P	S	Si	Cr	Ni	Mo	Other
329	.08	2.0	.040	.03	.75	23.0/ 28.0	2.5/ 5.0	1.0/ 2.0	
330	.08	2.0	.040	.03	.75/ 1.0	17.0/ 20.0	34.0/ 37.0		
347	.08	2.0	.045	.03	.75	17.0/ 19.0	9.0/ 13.0		10xC(min)/ 1(max)Cb+Ta
348	.08	2.0	.045	.03	.75	17.0/ 19.0	9.0/ 13.0		10xC(min)/ 1(max)Cb+Ta .1Ta,0.20Co
384	.08	2.0	.045	.03	1.0	15.0/ 17.0	17.0/ 19.0		

(Source : AISI Designers Handbook Series)

This group of materials has substantial amounts of chrome and and nickel as its alloying elements. These elements provide corrosion resistance for the stainless steels shown. As the content of chromium (Cr) is increased in the austenitic stainless steels, so is the high temperature corrosion resistance of these materials. These materials are strengthened by adding nitrogen (N), and when sulphur (S) has been added, it enhances machinability as in the case of 303 stainless steel. The carbon (C) content is lowered in 304L and 316L to stabilize these materials following welding, while the ratio of chromium/nickel (Cr/Ni) in 301 and 305 stainless is changed in order to obtain different forming characteristics of these materials. In general, type 304 stainless steel is regarded as the base material in the austenitic group, and all remaining alloys within this designation are slight modifications of type 304.

PHYSICAL PROPERTIES OF AUSTENITIC STAINLESS STEELS

AISI#	Tens. Strgth. Minimum kpsi	Yield Strgth. Minimum kpsi	Ductil. % elongation/ 2"	Modulus of Elasticity mpsi	Thermal Conductivity BTU/ft/hr/F	Thermal Expansion @32-212F in/in/F
201	75	40	40	28.6	9.4	8.7
202	75	40	40	28.6	9.4	9.2
205	110	65	30	28.0	---	9.4
301	75	30	40	28.0	9.4	9.4
302	75	30	40	28.0	9.4	9.6
302B	•75	30	40	28.0	---	9.0

PHYSICAL PROPERTIES OF AUSTENITIC STAINLESS STEELS CONTINUED

AISI#	Tens. Strgth. Minimum kpsi	Yield Strgth. Minimum kpsi	Ductil. % elongation/ 2"	Modulus of Elasticity mpsi	Thermal Conductivity BTU/ft/hr/F	Thermal Expansion @32-212F in/in/F
303	85	35	50	28.0	9.4	9.6
303Se	85	35	50	28.0	9.4	9.6
304	75	30	40	28.0	9.4	9.6
304L	70	25	40	28.0	9.4	9.6
304Cu	65-85	30	70	28.0	9.4	9.6
304N	80	35	30	28.5	9.4	9.6
305	75	30	40	28.0	9.4	9.6
308	75	30	40	28.0	8.8	9.6
309	75	30	40	29.0	9.0	8.3
309S	75	30	40	29.0	9.0	8.3
310	75	30	40	29.0	8.2	8.8
310S	75	30	40	29.0	8.2	8.8
316	75	30	40	28.0	9.4	8.9
316L	75	30	40	28.0	9.4	8.9
316F	85	35	40	29.0	8.3	9.2
316N	80	35	30	28.5	---	8.9
317	75	30	40	28.0	9.4	8.9
317L	85	35	75	29.0	8.3	9.2
321	75	30	40	28.0	9.3	9.3
329	105	80	25	27.0	---	5.6
330	70	30	30	28.5	7.2	9.3
347	75	30	40	28.0	9.3	9.3
348	75	30	40	28.0	9.3	9.3
384	60-80	30	40	28.0	9.6	9.6

(Source : AISI Designers Handbook Series)

The austenitic stainless steels should be machined with positive rake cutting geometries for drilling, milling and turning operations. This is primarily warranted since materials which fall within this stainless steel classification have relatively large ductilities (30-75% elongation in 2"). This indicates that they can absorb a tremendous amount of strain (stretching under load), which increases their hardness. When negative geometries are used on these materials, strain hardening occurs, as well as significant welding of the chip to the cutting edge because of the heat and chip deformation generated during machining. The 200 & 300 series stainless steels should be machined with as positive a cutting geometry as possible (strong enough to avoid breakage) to minimize the workhardening condition, and to keep the chip actively moving along the cutting edge. For this reason, milling cutters with large (30 degrees or larger) lead angles and positive rake angles are preferred. In addition, turning tools with positive chip breaker configurations are also required on these materials to prevent chip welding and prevent hardening of the workpiece.

Since, by definition, austenitic stainless steels are hardened due to cold work, adequate chip loads must be used when machining these materials. If light chip loads (.0015" or less) are selected on 200 & 300 series, stainless steels excessive heat is produced due to relatively high frictional forces between the workpiece and cutting edge. This type of rubbing (as opposed to cutting) condition produces a burnished and, thus, hardened workpiece surface. Coolant is recommended on these materials to improve the lubricity between the part and cutting edge and to regulate any excessive heating of the part.

Ferritic Stainless Steels contain almost no nickel and less manganese than austenitic stainless steels. The ferritic group comprises most of the 400 series stainless steels, and they are primarily alloyed with chromium. This series of stainless steels has higher machinability ratings than the austenitic group. The ferritic group is hardened through heat treatment, not cold working, and, therefore, the ductility of these materials is lower than that of the 200 & 300 series stainless steels. The ductility and toughness of these materials can be controlled by altering the carbon and nitrogen contents. Reductions in these elements will lower ductility and toughness. The chemical compositions of the ferritic stainless steels are as follows:

CHEMICAL COMPOSITION OF FERRITIC STAINLESS STEELS

	Chemical Analysis % (Maximum unless noted otherwise)							
AISI#	C	Mn	P	S	Si	Cr	Ni	Mo	Other
405	.08	1.0	.040	.03	1.0	11.5/ 14.5			.10/ .30 Al
409	.08	1.0	.045	.045	1.0	10.5/ 11.75			6xC/ .75 Ti
429	.12	1.0	.040	.03	1.0	14.0/ 16.0	.75		
430	.12	1.0	.040	.03	1.0	16.0/ 18.0	.75		
430F	.12	1.25	.060	.15	1.0	16.0/ 18.0			
431	.20	1.0	.040	.03	1.0	15.0/ 17.0	1.25/ 2.50		
434	.12	1.0	.040	.03	1.0	16.0/ 18.0		.75/ 1.25	

CHEMICAL COMPOSITION OF FERRITIC STAINLESS STEELS CONTINUED

AISI#	C	Mn	P	S	Si	Cr	Ni	Mo	Other
	Chemical Analysis % (Maximum unless noted otherwise)								
436	.12	1.0	.040	.03	1.0	16.0/ 18.0		.75/ 1.25	5xC/ .7 Cb+Ta
439	.07	1.0	.040	.03	1.0	17.0/ 19.0	.50		.2+4(C+N) (min)Ti .15 Al
442	.20	1.0	.040	.03	1.0	18.0/ 23.0			
444	.025	1.0	.040	.03	1.0	17.5/ 19.5		.75/ 1.25	.035N
446	.20	1.5	.040	.03	1.0	23.0/ 27.0			.25N

(Source : AISI Designers Handbook Series)

As stated earlier, the nickel content of the ferritic stainless steels is minimal when compared to the austenitic group. In addition, these alloys have slightly higher contents of manganese (Mn) and carbon (C) when compared to the 200 & 300 series stainless steels. The physical properties of the ferritic group is as follows:

PHYSICAL PROPERTIES OF FERRITIC STAINLESS STEELS

AISI#	Tens. Strgth. Minimum kpsi	Yield Strgth. Minimum kpsi	Ductil. % elongation/ 2"	Modulus of Elasticity mpsi	Thermal Conductivity BTU/ft/hr/F	Thermal Expansion @32-212F in/in/F
405	60	25	20	29.0	15.6	6.0
409	65	35	25	29.0	14.4	6.5
429	70	40	20	29.0	14.8	5.7
430	70	40	20	29.0	13.8	5.8
430F	80	55	25	29.0	15.1	5.8
431	125	95	20	29.0	11.7	5.6
434	77	53	23	29.0	13.8	5.8
436	77	53	23	29.0	13.8	5.2
439	70	40	20	29.0	----	---
442	80	45	25	29.0	12.5	5.6
444	60	40	20	29.0	15.5	6.1
446	70	40	20	29.0	12.1	5.8

(Source : AISI Designers Handbook Series)

Ferritic stainless steels exhibit slightly lower average tensile strength levels than austenitic stainless steels. In addition, the average ductility of ferritic stainless steels is 22%, while austenitic steels average ductilities of 42%. These two physical properties hinder the machinability of 200 and 300 series stainless steels. Therefore, ferritic stainless steels are considered to have higher machinability levels. Positive cutting geometries are also required for this group of stainless steels: however, this group has a relatively low susceptibility to workhardening as compared to the austenitic group. The use of coolant on these materials is advised for lubricity as well as temperature control.

Martensitic Stainless Steels are also covered by the AISI 400 series designation. Martensitic, like ferritic stainless steels, have very little, if any, nickel content. This group of materials has slightly lower levels of chromium and higher levels of carbon when compared to ferritic stainless steels. The higher ratio of carbon to chromium content increases the hardenability of these materials when heat treated. This group of stainless steels is normally heat treated to attain their final hardnesses. The chemical content of these materials is shown below:

CHEMICAL COMPOSITION OF MARTENSITIC STAINLESS STEELS

	Chemical Analysis % (Maximum unless noted otherwise)								
AISI#	C	Mn	P	S	Si	Cr	Ni	Mo	Other
403	.15	1.0	.040	.03	.50	11.5/ 13.0			
410	.15	1.0	.040	.03	1.0	11.5/ 13.5	.75		
410S	.08	1.0	.040	.03	1.0	11.5/ 13.5	.60		
414	.15	1.0	.040	.03	1.0	11.5/ 13.5	1.25/ 2.50		
416	.15	1.25	.060	.15	1.0	12.0/ 14.0			
420	over .15	1.0	.040	.03	1.0	12.0/ 14.0			
420F	over .15	1.25	.060	.15	1.0	12.0/ 14.0			
422	.20/ .25	.50/ 1.0	.025	.025	.5	11.0/ 12.5	.50/ 1.0	.90/ 1.25	.2/.3V .9/1.25W

CHEMICAL COMPOSITION OF MARTENSITIC STAINLESS STEELS CONT.

AISI#	C	Mn	P	S	Si	Cr	Ni	Mo	Other
	Chemical Analysis % (Maximum unless noted otherwise)								
440A	.60/ .75	1.0	.040	.03	1.0	16.0/ 18.0		.75	
440B	.75/ .95	1.0	.040	.03	1.0	16.0/ 18.0		.75	
440C	.95/ 1.20	1.0	.040	.03	1.0	16.0/ 18.0		.75	

(Source : AISI Designers Handbook Series)

Typical physical properties of martensitic stainless steels are as follows:

PHYSICAL PROPERTIES OF MARTENSITIC STAINLESS STEELS

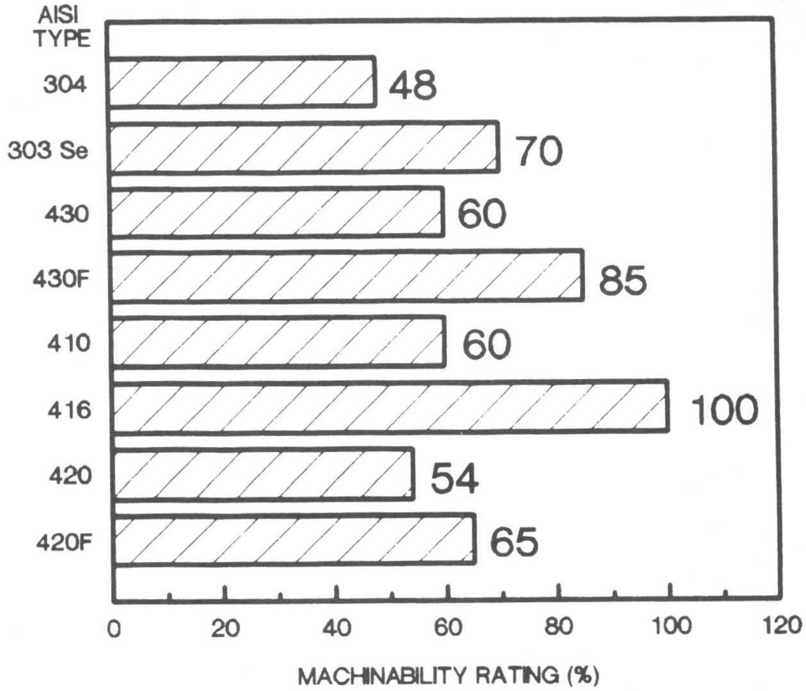
AISI#	Tens. Strgth. Minimum kpsi	Yield Strgth. Minimum kpsi	Ductil. % elon-gation/ 2"	Modulus of Elast-icity mpsi	Thermal Conduc-tivity BTU/ft/hr/F	Thermal Expansion @32-212F in/in/F
403	70	40	20	29.0	14.4	5.5
410	70	40	20	29.0	14.4	5.5
410S	70	35	30	29.0	14.4	5.5
414	115	90	20	29.0	14.4	5.8
416	75	40	30	29.0	14.4	5.5
420	95	50	25	29.0	14.4	5.7
420F	95	55	22	29.0	14.5	5.7
422	140	110	13	29.8	13.8	6.2
440A	105	60	20	29.0	14.0	5.7
440B	107	62	18	29.0	14.0	5.7
440C	110	65	14	29.0	14.0	5.7

(Source : AISI Designers Handbook Series)

The strength readings shown above are for stainless steel which are not in a fully hardened condition. The only heat treated specimen is the AISI 422 stainless steel and, therefore, it has a higher strength level than the other martensitic stainless steels shown. Since martensitic stainless steels are strengthened by heat treatment, they can exhibit tensile strengths up to 300,000 psi at 580 BHN, and, thus, the machinability of these alloys is totally dependent on their hardness.

The graph on page 33 indicates the relative machinability of stainless steel alloys. The scale of this graph is consistant with previous machinability ratings stated earlier.

MACHINABILITY OF STAINLESS STEELS



Martensitic stainless steels are considered to be very machinable in a non-heat treated condition. In fact, AISI 416 stainless steel is often considered to have a machinability rating equal to B1112 steel, the foundation on which these ratings are based. As stated earlier, positive geometries are recommended for martensitic stainless steels in a non-heat treated condition. When these materials are heat treated beyond 380 BHN, their ductility is significantly reduced. In these cases neutral or slightly negative geometries can be utilized, but chip welding to the cutting edge will still be a problem. As in previous discussions, coolant is recommended for these materials to reduce heat and increase lubricity.

Precipitation Hardening Stainless Steels are a small group of materials with a relatively high chromium and nickel content which are hardened by using an aging process. This process transforms the grain structure of the material and thus the final hardness of the stainless steel alloy. Normal numeral designations of precipitation hardened stainless steels are 17-4PH or 13-8PH. In these designations, the first number indicates the percentage of chromium (Cr) content, and the second number indicates the nickel (Ni) content of the alloy. For example, a 17-4PH stainless steel has 17% chromium and 4% nickel composition on average by weight and is treated or hardened by precipitation hardening (PH). This group of stainless steels has the same relative strength characteristics as the martensitic group for comparable hardness levels. The chemical composition of these alloys is as follows:

CHEMICAL COMPOSITION OF PH STAINLESS STEELS

AISI#	C	Mn	P	S	Si	Cr	Ni	Mo	Other
	Chemical Analysis % (Maximum unless noted otherwise)								
S13800 (13-8PH)	.05	.10	.010	.008	.10	12.25/ 13.25	7.5/ 8.5	2.0/ 2.5	.01N .9/1.35Al
S15500 (15-5PH)	.07	1.0	.040	.03	1.0	14.0/ 15.5	3.5/ 5.5		2.5/4.5Cu .15/.45Cb+Ta
S17400 (17-4PH)	.07	1.0	.040	.03	1.0	15.0/ 17.5	3.0/ 5.0		3.0/5.0Cu .15/.45Cb+Ta
S17700 (17-7PH)	.09	1.0	.040	.04	1.0	16.0/ 18.0	6.5/ 7.75		.75/1.5 Al

(Source : AISI Designers Handbook Series)

Although a range is given for both chromium and nickel contents of these materials, the part number designation is often an average or midpoint of the specified ranges. The physical properties of these materials are as follows:

PHYSICAL PROPERTIES OF PH STAINLESS STEELS

AISI#	Tens. Strgth. Minimum kpsi	Yield Strgth. Minimum kpsi	Ductil. % elongation/ 2"	Modulus of Elasticity mpsi	Thermal Conductivity BTU/ft/hr/F	Thermal Expansion @32-212F in/in/F
S13800 13-8PH	160	120	17	29.4	8.1	5.9
S15500 15-5PH	160	145	15	28.5	10.3	6.0
S17400 17-4PH	160	145	15	28.5	10.6	6.0
S17700 17-7PH	150	65	25	29.5	9.5	8.6

(Source : AISI Designers Handbook Series)

Stainless steel parts are precipitation hardened to attain higher strength levels at a relatively low temperature. These materials often exhibit the same hardness and strength levels and, thus, machining characteristics as those of the martensitic stainless steels. Therefore, the machining guidelines outlined for martensitic steels should be followed for the PH stainless steel alloys.

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 light-weight metal (1/3 the density) when compared to steel, yet it posseses 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 available as an engineering material in two distinct forms. First, pure aluminum (99.99%) is used in a wide variety of commercial and industrial applications, normally for its resistance to corrosion, reflective or heat conducting properties. The use of pure aluminum is relatively small when compared to aluminum alloys. When an application requires additional strength or a specific property enhancement, an alloyed aluminum is normally selected from a vast assortment of commercial grades. Alloyed aluminum consists of roughly 80-96% aluminum by weight mixed with a variety of other metallic elements. These alloyed aluminums can be wrought or cast alloys depending on the finished part configuration and the properties desired. Let's examine wrought and and cast aluminums individually:

WROUGHT ALUMINUM ALLOYS

Wrought aluminum alloys are covered by a four digit numerical code which is used to distinguish one grade from another by chemical composition. This code was developed and instituted by the Aluminum Association of America in 1954.

The first numeral in the four digit code usually indicates the major alloying element contained in the wrought alloy, except for the 1xxx series materials which are 99% pure aluminum. In all other cases the major alloying element in wrought aluminum alloys is detailed below:

WROUGHT ALUMINUM ALLOYS

Major Alloying Element	Designation
Copper (Cu)	2xxx
Manganese (Mn)	3xxx
Silicon (Si)	4xxx
Magnesium (Mg)	5xxx
Magnesium & Silicon	6xxx
Zinc (Zn)	7xxx
Other	8xxx

The second digit in the designation indicates either some sort of alloying of the product or a specific level of impurity. When the second digit in the wrought aluminum numbering system is zero, there is no particular control of impurities in the final product. Any other numeric designation in the second position of the part number indicates control of one or more impurities in the material. This numbering system is best explained by reviewing several individual cases with examples.

As stated earlier, the 1xxx wrought aluminum alloy group is 99% pure aluminum. The last two digits for this product group establish the level of pure aluminum present beyond 99%, in hundredths of a percent. For example, a 1050 wrought aluminum contains a minimum of 99.50% pure aluminum, with up to .50% of other impurities or alloying elements. The zero in the second position of this part number indicates that the remaining impurities in this material aren't controlled.

The last two numbers or positions in the 2xxx to 8xxx alloy designation have no specific meaning. Therefore, they are used primarily to separate specific wrought grades without explaining chemical makeup or content.

As stated earlier, pure aluminum is alloyed in order to improve its strength. Thus, the wrought aluminums have higher strength levels than a commercially pure aluminum, even in an unhardened state. But, frequently it is necessary to increase the strength of wrought aluminums to meet the requirements of individual material applications (i.e.) aircraft structural parts. Wrought aluminum alloys are usually separated into two distinct groups; those which are strengthened by cold working (non-heat-treatable) and those which can be hardened and strengthened through heat treatment (heat-treatable). The following list shows the wrought designations which fall into each of these categories:

WROUGHT ALUMINUM ALLOYS

Non-Heat-Treatable	Heat Treatable
1000 series	2000 series
3000 series	6000 series
4000 series	7000 series
5000 series	8000 series

(Source : Machinery's Handbook)

The non-heat-treatable alloys are strengthened through cold working. The level of cold work of a particular alloy is indicated by its temper ("H") which follows the series number and is separated by a hyphen. For heat-treatable alloys, increased strength is attained through the heating of the part which is often called solution heat treatment and is designated by a "T", which also follows the series number and is separated by a hyphen. A definition of some of the nomenclature associated with these hardening processes is as follows:

F - As Fabricated

O - Annealed (Wrought Products Only)

H - Strain Hardened (Wrought Products Only)

H1 - Strain Hardened Only
H2 - Strain Hardened and Partially Annealed
H3 - Strain Hardened and Stabilized

W - Solution Heat Treated (Unstable Temper)

T - Solution Heat Treated (Stable Temper)

T1 - Naturally Aged
T2 - Annealed (Castings Only)
T3 - Solution Heat Treated and Cold Worked
T4 - Solution Heat Treated & Naturally Aged
T5 - Artificially Aged Only
T6 - Solution Heat Treated & Artificially Aged
T7 - Solution Heat Treated & Stabilized
T8 - Solution Heat Treated, Cold Worked & Then Artificially Aged
T9 - Solution Heat Treated, Artificially Aged & Then Cold Worked
T10 - Artificially Aged & Cold Worked

(Source : Machinery's Handbook)

Machinery's Handbook details many additional temper ("H") and solution treatment ("T") designations beyond those shown in the above list. The following chart gives a brief idea of how different tempers and solution treatments influence mechanical material properties:

TYPICAL MECHANICAL PROPERTIES OF WROUGHT ALUMINUM ALLOYS

Aluminum Alloy & Temper Or Solution Treatment	Yield Strength (Psi)	Tensile Strength (Psi)	Modulus of Elasticity (Psi)	Hardness (BHN)
1060-0	4,000	10,000	10,000,000	19
1060-H18	11,000	19,000	10,000,000	35
2014-0	14,000	27,000	10,000,000	45
2014-T6	60,000	70,000	10,000,000	135
2025-T6	37,000	58,000	10,000,000	110
2117-T4	24,000	43,000	10,000,000	70
3004-0	10,000	26,000	10,000,000	45

TYP. MECHANICAL PROPERTIES OF WROUGHT ALUMINUM ALLOYS CONT.

Aluminum Alloy & Temper Or Solution Treatment	Yield Strength (Psi)	Tensile Strength (Psi)	Modulus of Elasticity (Psi)	Hardness (BHN)
3004-H38	36,000	41,000	10,000,000	77
6061-O	8,000	18,000	10,000,000	30
6061-T6	40,000	45,000	10,000,000	95
7075-O	15,000	33,000	10,000,000	60
7075-T6	73,000	83,000	10,000,000	150

(Source : Machinery's Handbook)

Wrought aluminum alloys generally have extremely high machinability ratings, since these materials are of a relatively low strength and hardness. The only detrimental machining characteristic of these materials is their tendency to weld or build up along the edge (BUE) of HSS and Carbide cutting tools. This condition is improved or eliminated by using highly positive cutting geometries (rake angles in excess of 20 degrees), adequate speed (minimum of 2000 SFPM), sufficient feed rates (minimum of .005 IPR) and flood coolant (water soluble). Highly positive cutting geometries tend to shear or slice the wrought aluminum alloys during machining operations. As cutting geometries become less positive, cutting forces and heat increase and the aluminum flows less freely across the rake face of the cutting tool and, therefore, welding and built-up edge results. In addition, wrought aluminum alloys are very elastic, and, thus, they deflect at three times the rate of steel when subjected to the same load. This is another essential reason for the use of low force, sharp, high shear cutting geometries on wrought aluminum alloys. T-landed inserts should be used on these materials only when excessive chip thicknesses prematurely fracture or micro-chip sharp edged inserts.

CAST ALUMINUM ALLOYS

Cast aluminum alloys are designated by a three digit ANSI code. The main difference between the various cast alloy groups is the major alloying element contained in the casting. In addition, the three digit code is often followed by a decimal point and a single additional digit following the decimal. The first digit in this numbering system identifies the major alloying element in the cast part, just as it was in the case of the wrought aluminum alloys. The remaining digits have no specific meaning and are, therefore, used to separate different alloys. The numerical position following the decimal point is normally either a 0, 1 or 2 and indicates the actual form of the product. Cast alloys utilize the same temper and solution treatment nomenclature and designations outlined on page 35. The group 1xx is composed of alloys that contain 99% pure aluminum. In all of the remaining cases the listing of the major alloying elements in cast aluminum alloys is detailed below:

CAST ALUMINUM ALLOYS

Major Alloying Element	Designation
Copper (Cu)	2xx
Silicon + Copper and/or Manganese	3xx
Silicon (Si)	4xx
Magnesium (Mg)	5xx
Not in current use	6xx
Zinc (Zn)	7xx
Tin (Sn)	8xx
Others	9xx

The machinability ratings of cast aluminum alloys is often as high as the wrought materials. Hardness levels of tempered cast alloys are normally less than 130 Brinell. Therefore, generally speaking, the guidelines in terms of cutting geometry, speed and feed rate recommended for wrought alloys apply to cast alloys. The only exceptions to these guidelines are those cast alloys with significant silicon contents (in excess of 14%). The ANSI cast aluminum grades 390 and 392 have silicon contents of 17-19% and, thus, these materials produce significant edge wear on many cutting tools due to the hard abrasive silicon particles dispersed in the aluminum. For this reason, stronger less positive (approximately 10 degree) rake face geometries are recommended and are necessary to combat the higher cutting forces produced by these materials. In addition, premature edge wear can be avoided by running much more moderate cutting speeds (500 SFPM) on high silicon aluminums.

COPPER

Copper is a very popular material which is widely used for its superior electrical conductivity, corrosion resistance and ease in formability. In addition, when alloyed properly, copper alloys can exhibit a vast array of strength levels and unique mechanical properties. Copper is available in wrought shapes as well as castings. Wrought copper products include: sheets, pipe, wire and rod.

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

BRASS

Brass is the most popular commercial copper alloy. This material is available in many different forms, but its main alloying elements are always some combination of copper and zinc. As the zinc content of brass is increased, the resulting materials change in color. The brass materials with less than 15% zinc are of a reddish color, while those which exceed 24% are yellow. The addition of lead and zinc generally improves the machining characteristics of these materials. These materials are available as either cast or wrought parts. Brass is a popular material for pipe fittings, electrical terminals, valves and bushings.

BRONZE

Bronze is a stronger, tougher and less machinable copper alloy than brass. Tin, aluminum, nickel, silicon and beryllium are often used as alloying elements to improve the strength and introduce different mechanical properties in bronze alloys. Bronze like brass, is available as a wrought or cast alloy. This material has wide spread use in castings for the marine industry, valve stems, gears, brackets, bushings and bearings. Bronze is a more expensive material than brass and, therefore, price as well as poor machining characterisitics limit the scope of its application as an engineering material.

The classification for copper alloys is now covered by the UNS (United Numbering System) code. Individual alloy designations identified by this code begin with the letter "C" followed by five digits. The outline of the entire numbering system for wrought copper and its alloys is presented on the next page:

UNS CLASSIFICATION OF WROUGHT COPPER & COPPER ALLOYS

Family	Major Alloy Element	UNS Designation
Coppers, High copper alloys	-	C1xxxx
Brasses	Zn	C2xxxx, C3xxxx, C4xxxx, C66400-C69800
Phosphor bronzes	Sn	C5xxxx
Aluminum bronzes	Al	C60600-C64200
Silicon bronzes	Si	C64700-C66100
Copper nickels, nickel silvers	Ni	C7xxxx

(Source : Machinery's Handbook)

The UNS specification alloys includes several specific designations to cover the vast assortment of commercial cast copper alloys. A relatively comprehensive summary of UNS cast copper alloys is shown below:

UNS CLASSIFICATION OF CAST COPPER & COPPER ALLOYS

Family	Major Alloy Element	UNS Designation
Coppers, High copper alloys	-	C80000-C82800
Red brasses & leaded bronzes	Pb,Zn	C83300-C83800
Semi-red brasses	Pb,Zn	C84200-C84800
Yellow brasses	Pb,Zn	C85200-C85800
Manganese bronze	Mn,Zn	C86100-C86800
Silicon bronze & brass	Si,Zn	C87200-C87900
Tin bronze	Sn,Zn	C90200-C91700
Leaded tin bronze	Pb,Sn,Zn	C92200-C94500
Nickel-tin bronze	Ni,Sn,Zn	C94700-C94900
Aluminum bronze	Al	C95200-C95800
Copper nickels	Ni	C96200-C96700
Nickel silver	Ag	C97300-C97800
Leaded copper	Pb	C98200-C98800
Special alloys	Many	C99300-C99750

(Source : Machinery's Handbook)

The machinability of wrought and cast copper and its alloys varies widely depending on the major alloying element in the UNS material. In general, copper alloys are regarded as being very tough materials. Brasses, leaded bronzes, highly leaded tin bronzes and nickel silvers are the only real exceptions to this rule. The machinability of wrought and cast copper alloys is outlined on the following page by UNS Designation:

MACHINABILITY OF WROUGHT & CAST COPPER & COPPER ALLOYS

Family	UNS Designation	Machinability Rating
Coppers, High copper alloys	C1xxxx	10-30%
Brasses	C2xxxx-C4xxxx	20-100%
	C66400-C69800	25-30%
Phosphor bronzes	C5xxxx	20%
Aluminum bronzes	C60600-C64200	20-60%
Silicon bronzes	C64700-C66100	30%
Copper nickels, nick. silvers	C7xxxx	20-60%
Coppers, High copper alloys	C80000-C82800	10-30%
Red brasses & leaded bronzes	C83300-C83800	35-90%
Semi-red brasses	C84200-C84800	80-90%
Yellow brasses	C85200-C85800	80%
Manganese bronze	C86100-C86800	8-65%
Silicon bronze & brass	C87200-C87900	40-80%
Tin bronze	C90200-C91700	10-20%
Leaded tin bronze	C92200-C94500	30-80%
Nickel-tin bronze	C94700-C94900	30-50%
Aluminum bronze	C95200-C95800	50-60%
Copper nickels	C96200-C96700	10-20%
Nickel silver	C97300-C97800	60-70%
Leaded copper	C98200-C98800	unknown
Special alloys	C99300-C99750	20-80%

The machinability of copper and its alloys varies widely. Pure copper and high copper alloys are very tough, abrasive and prone to tearing. This set of characteristics limits their machinability significantly, as indicated by the chart shown above. 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. However, care must be taken in the application of tools on these materials, since they can fracture cutting edges based on their extreme strength. Negative axial and positive radial rake angle geometries should be used on semi-red and yellow 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 (Waspaloys), 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, Incoloys, and several Haynes alloys. The machinability of nickel based alloys is generally quite low. However, there are some alloys which reach 25% ratings. Nickel alloys are available in wrought or cast form, heat-treatable or non-heat-treatable. Non-heat-treatable nickel based materials can be cold worked to extreme hardnesses and strengths. Many of the nickel based alloys are designated using either the UNS or SAE numerical code. However, in general machining practice, these alloys are often referred to by their trade names and, therefore, we will not deviate from this practice.

INCONEL

Inconel is a nickel-chromium alloy which is often utilized for its excellent oxidation resistance and superior strength from extremely low to elevated temperatures. This material is a popular choice in the nuclear and aerospace industries for many high temperature applications. In addition, special grades of Inconel are used for their corrosion resistance properties in the processing of acids and other chemicals. As stated earlier, many of these special grades of Inconel are referred to by their trade names instead of the actual SAE or ANSI designations. A summary of three of the most popular grades of Inconel in terms of their properties and main applications is shown below:

Inconel Alloy	Description Of Alloy	Main Application
600	High nickel and chromium alloy, for extremely corrosive environments at elevated temperatures.	Nuclear steam generator tubing, food & chemical processing equipment.
625	Extreme strength and toughness from sub zero temperatures to 1800 degrees F, along with corrosion and fatigue strength.	Nuclear reactors, chemical and pollution control equipment.
718	Impressive strength characteristics from -423 to 1300 degrees F, with resistance to oxidation up to 1800 degrees F.	Jet engines, rocket motors and nuclear fuel element spacers.

HASTELLOY

Hastelloy is considered a nickel-chromium-molybdenum alloy with a variety of unique properties which make it an excellent material choice under severe operating conditions. This material group is noted for its ability to withstand cyclical stress and temperature for thousands of hours without cracking or fatiguing. In addition, Hastelloys are noted for their oxidation resistance at elevated temperatures (in excess of 2000 degrees F). These materials can be readily heat-treated, welded and fabricated into a vast array of final products. Again, we will refer to the various Hastelloy grades by their trade names instead of their SAE or ANSI designations. The summary of the two most popular Hastelloy grades along with applications and descriptions is shown below:

Hastelloy Alloy	Description Of Alloy	Main Application
S	This high temperature alloy displays superior thermal stability as well as a moderate thermal expansion coefficient. It is designed to withstand cyclical temperature loading without failure.	This material is often used to make seal rings for gas turbine engines.
X	This alloy has shown superior resistance to stress corrosion cracking at elevated temperatures (1200-1400 degrees F) for extended periods of time (up to 16,000 hours) in petro-chemical applications.	Gas turbine engine components, furnace rolls, and chemical process equipment.

The chemical breakdown of some of the more popular nickel based alloys are as follows:

CHEMICAL COMPOSITION OF NICKEL ALLOYS

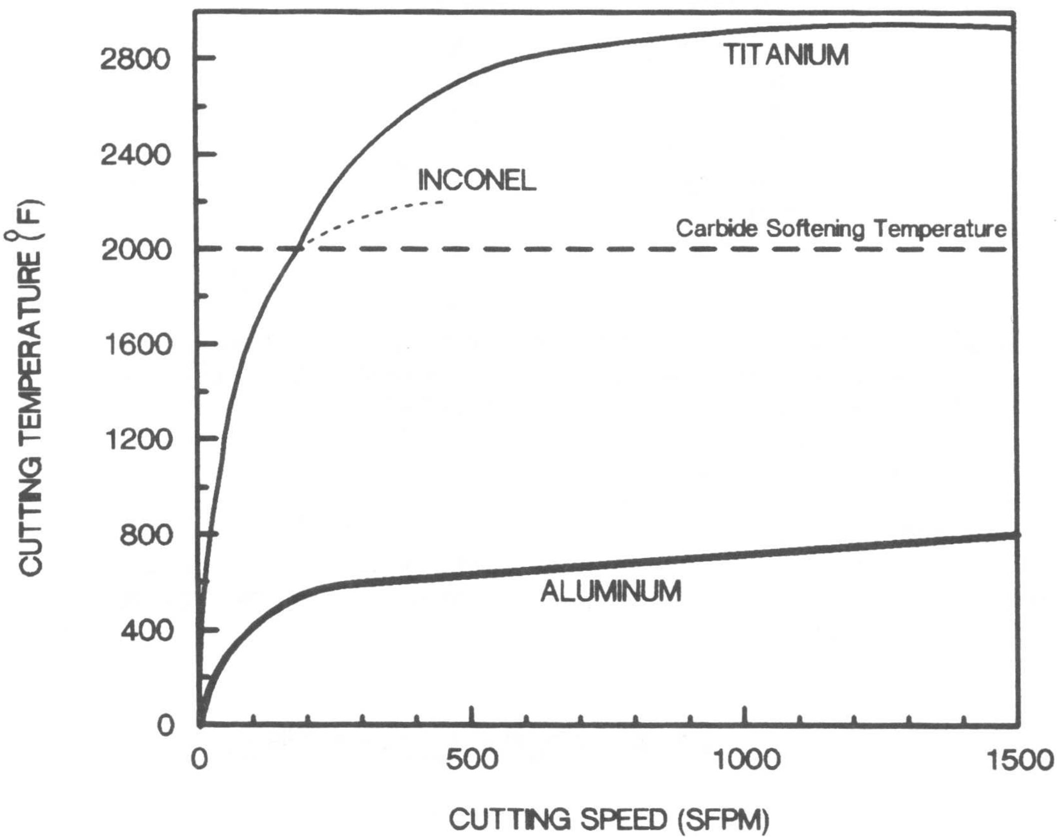
Alloy	Chemical Analysis % Ni & Co	Fe	Cr	Other	Typical Applications
Monel 400	66.5	1.2	-	31.5 Copper	heat exchangers
Monel K-500	66.5	1.0		29.5 Copper 2.7 Alum.	pump impellers, blades & shafts
Hastelloy X	48.0	19.0	22.0	9.0 Moly.	turbine combusters
Hastelloy S	65.7	3.0	15.5	14.5 Moly.	jet engine seals
Inconel 600	76.0	8.0	15.5	-	nuclear steam gen.
Inconel 625	61.0	2.5	21.5	9.0 Moly.	nuclear reactors
Inconel 718	52.5	18.5	19.0	3.0 Moly.	jet engines
Rene 41	60.8	5.0	19.0	9.8 Moly.	turbine blades
Waspaloy	70.0	3.0	19.5	4.3 Moly.	jet engine discs

(Source: Metal Progress)

As indicated by the chart, nickel based alloys are generally used in jet engine components and nuclear applications since they retain their strength and resist corrosion at elevated operating temperatures. Although thermal stability is a desired property of these materials in terms of their practical application, it creates significant problems when they are machined.

You might recall that machining occurs when an extremely hard (relative to the workpiece) cutting tool is used to remove material from a workpiece using either linear, rotational or linear and rotational motion. Machining cannot take place without either a differential in hardness or relative motion, between the tool and workpiece. As cutting speeds are increased during this process, both the tool and the workpiece begin softening in the cutting zone due to increased temperature. The ideal cutting condition is to select a speed (SFPM) where significant softening of the material begins without adversely effecting the hardness or normal wear rate of the tool. When machining alloys with high concentrations of nickel, the role of the carbide cutting tool is compromised in terms of the rate at which hardness, and thus strength, is lost due to increased cutting speed. This is illustrated on the graph shown on page 43.

MACHINING TEMPERATURE



The cutting speed used to machine nickel alloys with carbide tooling is generally low (under 120 SFPM), since carbide and Inconel have about the same melting or softening point. Therefore, as speeds and temperatures increase, carbide loses its hardness and strength at about the same rate as nickel alloys. This phenomenon limits the machinability of nickel based alloys which are presented below:

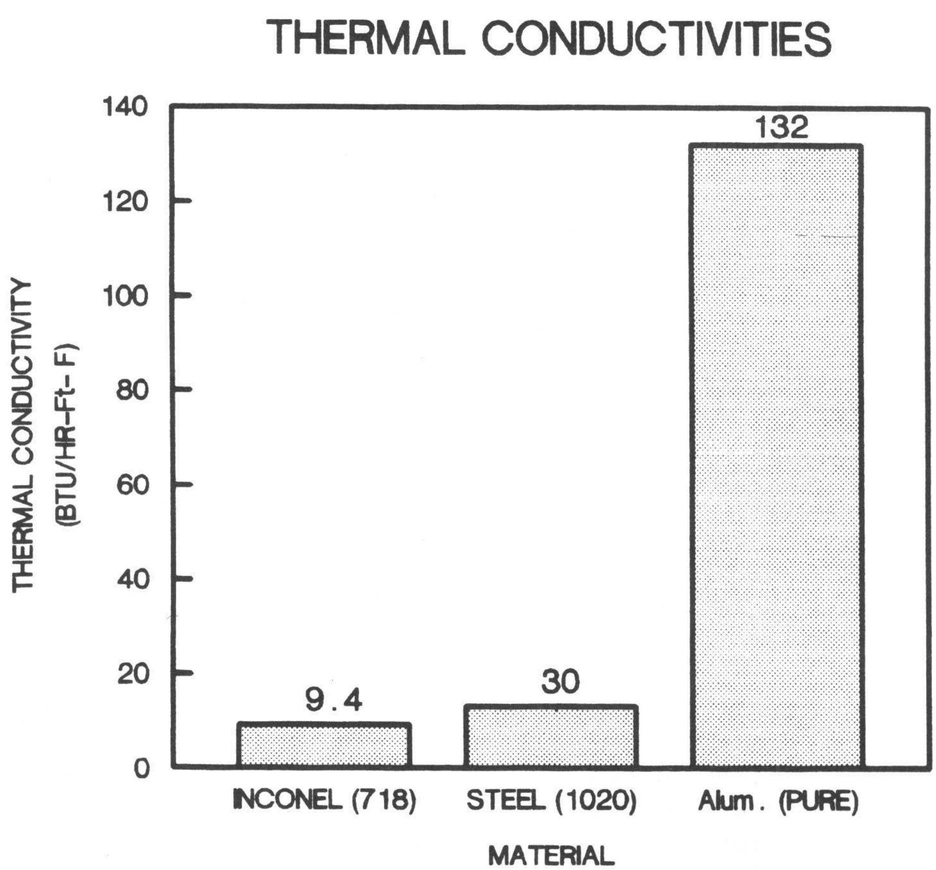
THE MACHINABILITY RATINGS OF NICKEL ALLOYS

Alloy	Machinability Rating
Astroloy	9%
Hastelloy X	18%
Inconel 600	11%
Inconel 625	11%
Inconel 706	16%
Inconel 718	12%

THE MACHINABILITY RATINGS OF NICKEL ALLOYS CONT.

Alloy	Machinability Rating
Inconel 903	17%
MA 754	24%
Rene 41	8%
Rene 77c	4%
Rene 80	4%
Udimet 700	6%
Waspaloy	15%

In addition, machinability is further reduced on these materials due to their relatively low thermal conductivities when compared to steels and aluminum alloys. This simply means that high nickel alloys dissipate heat very slowly and as they are machined heat tends to build up in the workpiece, cutting edge, and cutter body. Since excessive temperatures accelerate the wearing of the cutting edge, tool life will suffer. Obviously, as the carbide cutting speed is increased, this condition is amplified. See the graph of thermal conductivities shown below:



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. High nickel alloys have great heat resistance and a low thermal conductivity; thus, increased cutting speed is often more detrimental to the carbide cutting tool than it is an aid in softening these workpiece materials for improved productivity. Therefore, the machining of nickel alloys involves mechanical shearing with positive rake tools rather than molten metal shearing (high temperature) with tools operating at elevated cutting speeds. Adequate clearance angles must be utilized on these materials, since many of them are very ductile and prone to workhardening. When a cutter and nickel alloy part is in contact, the tool should always be moving. 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. This group of materials is available in one of the following three primary structural forms:

TITANIUM STRUCTURE	ALLOYING ELEMENTS
Alpha	Aluminum, Tin and Molybdenum
Alpha Beta	Aluminum, Tin , Vanadium and Molybdenum
Beta	Aluminum, Tin, Vanadium, and Chromium

Commercially pure titanium (99% in purity) alloys are often found in the chemical and nuclear industries based on their corrosion and oxidation resistance properties at elevated temperatures. This group of alloys are normally cold worked and annealed to their final strength and hardness levels. However, titanium alloys are periodically subjected to heat treatment to attain additional hardness and strength. In general, commercially pure titanium alloys have higher machinability when compared to the alpha, alpha beta and beta alloys due to their lower strength, hardness and ductilities. However, as a group, titanium and its alloys are often considered to have relatively poor machinability. This is due primarily to the poor tool life and productivity often encountered when machining these materials. The mechanical properties for titanium and its alloys are listed below:

TYP. MECHANICAL PROPERTIES OF TITANIUM AND TITANIUM ALLOYS

Titanium Designation	Yield Strength (Psi)	Tensile Strength (Psi)	Modulus of Elasticity (Psi)	Hardness (BHN)
99.5% Pure Ti	35,000	48,000	14,900,000	120
99.2% Pure Ti	50,000	63,000	14,900,000	200
99.1% Pure Ti	65,000	75,000	15,000,000	225
99.0% Pure Ti	85,000	96,000	15,100,000	265
Alpha Alloys				
Ti-5Al-2.5Sn	117,000	125,000	16,000,000	Rc36
Ti-5Al-2.5Sn ELI	108,000	117,000	16,000,000	Rc35
Ti-8Al-1Mo-1V	138,000	145,000	18,000,000	Rc35
Ti-6Al-2Sn-4Zr-2Mo	130,000	142,000	16,500,000	Rc32

TYP. MECH. PROPERTIES OF TITANIUM AND TITANIUM ALLOYS CONT.

Titanium Designation	Yield Strength (Psi)	Tensile Strength (Psi)	Modulus of Elasticity (Psi)	Hardness (BHN)
Alpha Beta Alloys				
Ti-3Al-2.5V	85,000	100,000	15,000,000	-
Ti-6Al-4V	134,000	144,000	16,500,000	Rc36
Ti-6Al-6V-2Sn	120,000	130,000	16,500,000	Rc35
Beta Alloys				
Ti-13V-11Cr-3Al	170,000	177,000	14,700,000	-
Ti-3Al-8V-6Cr-4Mo-4Zr	200,000	210,000	15,300,000	Rc42

(Source : Metal Progress)

As detailed in the previous designations, the actual percentage of alloying element contained in a particular alpha, alpha beta or beta titanium alloy is expressed in the actual numerical designation of the alloy. For example, the alpha beta alloy Ti-6Al-4V contains 6% aluminum, 4% vanadium and the remaining balance of the alloy is titanium. Alpha alloy Ti-6Al-2Sn-4Zr-2Mo also contains 6% aluminum, 2% tin, 4% zirconium, 2% molybdenum and the balance is titanium. Alloying elements are used to strengthen and harden titanium materials for more demanding applications. The following list below shows typical applications of commercially pure titanium and its alloys:

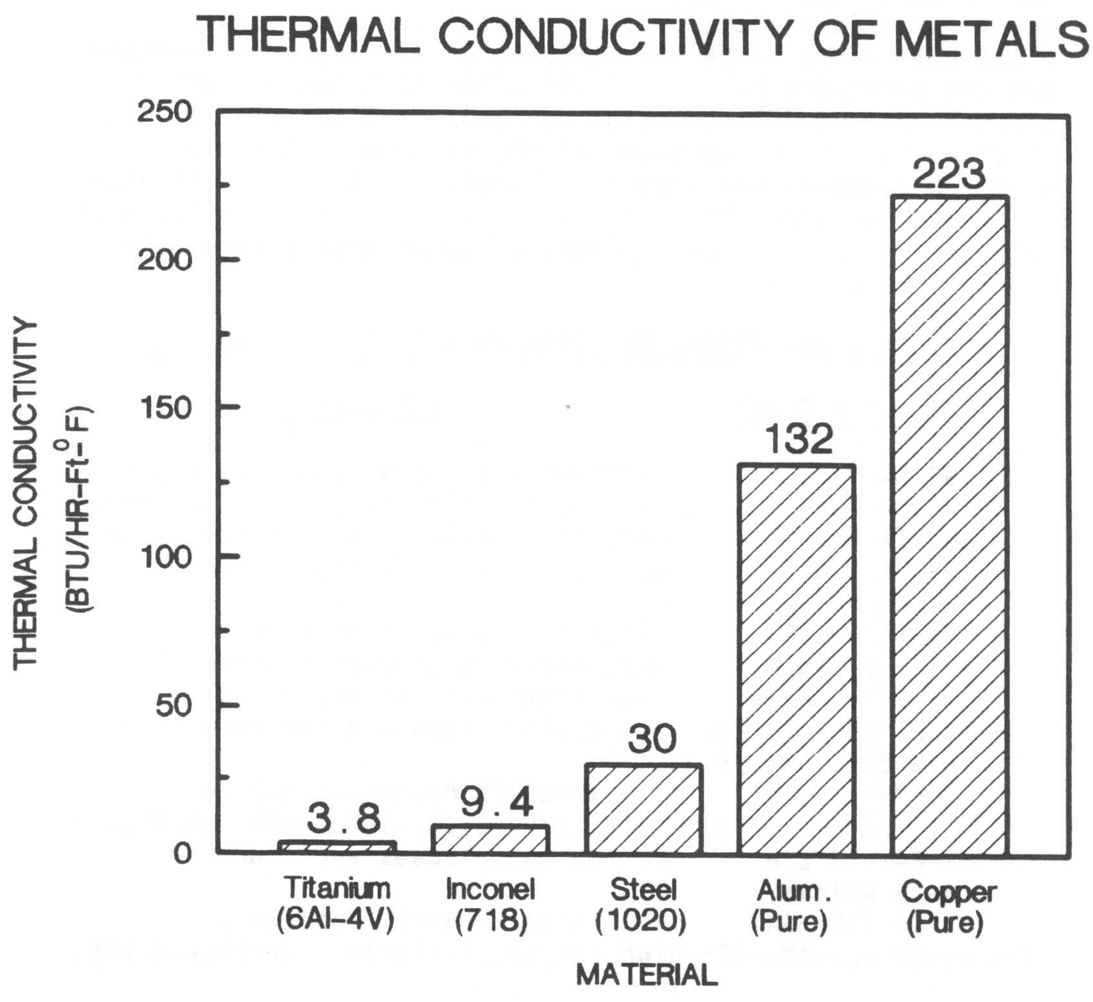
TYPICAL APPLICATIONS OF TITANIUM AND TITANIUM ALLOYS

Titanium Grade	Application
99.5% Pure Ti	airframes, chemical and marine parts
99.2% Pure Ti	marine parts, airframes, aircraft engs.
99.1% Pure Ti	chemical, marine and airframe parts
99.0% Pure Ti	surgical implants, high speed fans
Alpha Alloys	
Ti-5Al-2.5Sn	aircraft engine compressor blades
Ti-5Al-2.5Sn ELI	high pressure cryogenic vessels
Ti-8Al-1Mo-1V	airframe and jet engine parts
Ti-6Al-2Sn-4Zr-2Mo	jet engine cases and airframe skins
Alpha Beta Alloys	
Ti-3Al-2.5V	aircraft hydraulic tubing
Ti-6Al-4V	aircraft turb. blds. and discs, airframes
Ti-6Al-6V-2Sn	rocket motor cases and ordnance
Beta Alloys	
Ti-13V-11Cr-3Al	high strength fasteners
Ti-3Al-8V-6Cr-4Mo-4Zr	high strgth. fasteners, aerospace pts.

(Source : Metal Progress)

Pure titanium and its alloys have several unique physical properties which hinder their machinability. These materials have relatively high melting points, which limit the ultimate cutting speed and, thus, productivity at which they are machined. Carbide melts at a lower temperature level than titanium; therefore, if cutting speed is elevated to produce extreme temperatures in the cutting zone, welding will occur between the chip formed and the cutting edge, and particles of the cutting tool will be carried away with the chip. This phenomenon is known as "cratering", since an indentation or crater is formed in the tool rake face. Therefore, moderate cutting speeds are utilized on titanium alloys to avoid this condition, which reduces their actual machinability ratings.

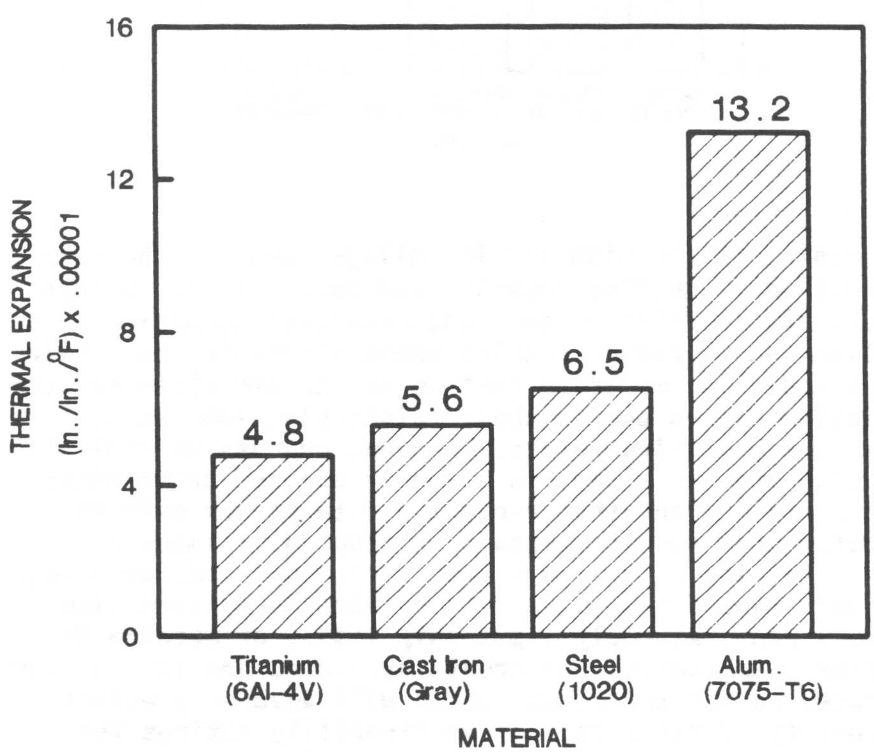
In addition, titanium has a relatively low thermal conductivity when compared to many other metals and metal alloys, thus, heat isn't dissipated from the cutting zone and cutting edge wear accelerates due to the elevated temperatures.



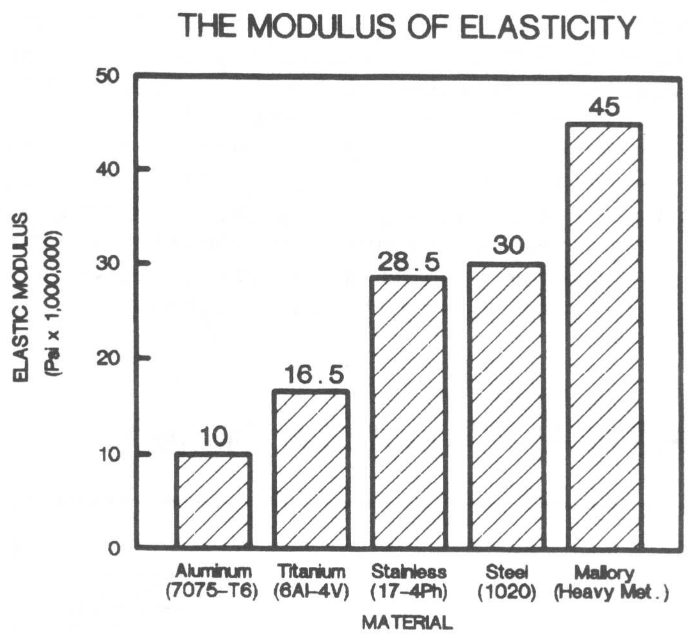
Titanium alloys are normally machined after annealing. The resulting hardness of these materials, after annealing, remains relatively high, which also reduces their ultimate machinability. As stated earlier, cutting speeds are reduced when machining hard parts to minimize cutting temperatures and thus edge wear.

The thermal expansion coefficient of titanium and its alloys is low compared to many other metals. Although this is a beneficial property in terms of maintaining dimensional stability of the workpiece, it is often detrimental in drilling operations. Efficient drilling operations in titanium require excellent coolant delivery and dispersion systems. If coolant isn't present in volume in the cutting zone, heat and temperature begin to rise and the tool will often expand more rapidly than the titanium part. Eventually, the drill begins producing a larger hole diameter at the hole bottom when compared to the initial hole size. If the drill is allowed to penetrate further into the part, this situation is magnified and a point is reached when the expanded drill cannot be withdrawn from the part without galling the hole or breaking the tool. This situation is obviously detrimental to the machinability of titanium, and it also makes deep hole drilling of this material a significant challenge.

THERMAL EXPANSION OF METALS



Titanium alloys also have extremely high chemical reactivities with other metallic substances. Therefore, when these materials are in the presence of other metals at an elevated temperature (i.e. cutting zone), they will tend to bond or weld at a very high rate. In the case of a carbide cutting tool, this welding or bonding means either built up edge, or rapid crater wear of the cutting edge, as the titanium chips carry minute carbide particles away from the cutting zone. Titanium is a very elastic material. Therefore, when subjected to external force, titanium will bend and deflect at twice the rate of steel. This tendency to move under load is a result of titanium's relatively low modulus of elasticity as illustrated below:



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. In addition, moderate cutting speeds are often selected for titanium alloys due to their relatively high chemical reactivity and melting points. The generous use of coolants on titanium and its alloys is strongly advised to maintain thermal stability and thus avoid the disasterous effects of accelerated heat and temperature buildup which leads to workpiece galling or tool breakage (drilling) and rapid edge wear. Tools should always be feeding when in contact with titanium parts. Extremely light chip loads, or machine drilling cycles with dwells, can create burnished and thus hardened surfaces on the workpiece, which will adversely effect the machinability of the part. The machinability ratings 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. A chemical breakdown of some of the more prominent refractory alloys is shown below:

CHEMICAL COMPOSITION OF REFRACTORY ALLOYS

	Chemical Analysis %											
Alloy	W	Mo	Cb	Ta	Hf	Zr	Ti	Ni	Fe	Cu	Re	C
Tungsten												
GE-218	bal	-	-	-	-	-	-	-	-	-	-	-
Anvilloy1100	bal	4	-	-	-	-	-	4	2	-	-	-
Gyromet	bal	1	-	-	-	-	-	3	1	7	-	-
Tantalum												
Ta-10W	10	-	-	bal	-	-	-	-	-	-	-	-
T-111	8	-	-	bal	2.4	-	-	-	-	-	-	-
T-222	9.6	-	-	bal	-	-	-	-	-	-	-	-
Molybdenum												
Mo	-	99	-	-	-	-	-	-	-	-	-	-
Mo-50Re	-	bal	-	-	-	-	-	-	-	-	50	-
Columbium												
C103	-	-	bal	-	10	.5	1	-	-	-	-	-
C129Y	10	-	bal	-	10	-	-	-	-	-	-	-

These materials are used in the aircraft and nuclear industries for their temperature and corrosion resistance. Tungsten and tantalum are used in the manufacture of sintered carbide cutting tools, while molybdenum is often alloyed with steel to improve its hardenability.

Cast molybdenum has a machinability rating of approximately 30%, while pure tungsten has a rating of only 5%. 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.

CHART OF METALLIC ELEMENTS

ELEMENT	SYMBOL
Aluminum	Al
Antimony	Sb
Arsenic	As
Barium	Ba
Beryllium	Be
Bismuth	Bi
Boron	B
Cadmium	Cd
Calcium	Ca
Carbon	C
Cerium	Ce
Chromium	Cr
Cobalt	Co
Columbium	Nb
Copper	Cu
Gallium	Ga
Germanium	Gw
Iron	Fe
Lead	Pb
Lithium	Li
Magnesium	Mg
Manganese	Mn
Molybdenum	Mo
Nickel	Ni
Phosphorous	P
Platinum	Pt
Rhodium	Rh
Selenium	Se
Silicon	Si
Sulphur	S
Tantalum	Ta
Tellurium	Te
Thallium	Tl
Thorium	Th
Tin	Sn
Titanium	Ti
Tungsten	W
Vanadium	V
Zinc	Zn
Zirconium	Ze

SUMMARY

The selection of the appropriate cutting geometry for a particular metal and machining operation is a function of the material hardness as well as its physical properties. Those materials which are extremely elastic, such as titanium and aluminum, normally require positive cutting geometries to reduce cutting forces and, thus, part deflection as they are machined. In addition, these materials tend to have high chemical reactivites, and positive geometries will minimize the heat and pressure produced during turning and milling operations, which will reduce built up edge on the tool. Cast iron parts are frequently milled and turned using negative rake angle tools due to the short chipping nature of the material, which produces concentrated loads on the outermost point along the cutting edge. However, since steels produce continuous chips, cutting loads are distributed across the rake face of the tool, and more positive cutting geometries are utilized as long as the material hardness is within the 20-40 Rockwell "c" range. Once hardness exceeds this level, neutral or negative rake carbide cutting geometries are required to absorb the increased impact loads and heat associated with this higher hardness condition on steel. Inconel and other heat resistant high nickel alloys are drilled, turned and milled with positive rake tooling since chip formation in these alloys is a function of mechanical shearing not heat induced shearing. Positive geometry is selected on this group of materials instead of cutting speed to promote efficient machining, since, as the temperature rises in the cutting zone, there is a more adverse effect on the strength of the carbide cutting edge than there is on the workpiece material. In addition, Inconel has a relatively low thermal conductivity. Therefore, heat tends to build in the cutting zone, which promotes rapid edge wear. Positive rake tools produce lower cutting temperatures and this is another reason why they are the appropriate choice for nickel based alloys.

It is obvious that several factors, beyond workpiece hardness, influence the selection of cutting tool geometries. The thorough understanding of the unique peculiarities from one material to another will assist in the screening of the appropriate cutting tool geometry for the workpiece material properties and material condition under consideration. The careless evaluation and review of the workpiece material condition will often lead to the misapplication of cutting tool geometry adversely influencing productivity and tool life. In other words, uninformed decision making in the cutting tool geometry selection process will frequently make the difference between success or failure, or at the very least, it will lead to a needless waste of time and money by increasing the numbers of demonstrations needed to arrive at the correct cutting tool solution.

INDEX OF TERMS

REVIEW QUESTIONS

1. How are the physical conditions of metals related to their machinability?

2. What is machinability?

3. Give three reasons why alpha-beta titanium alloys are difficult to machine.

4. Why are cutting speeds and feeds altered as the hardness of material increases or decreases?

5. Why is geometry selection important on very elastic materials?

6. How does a change in carbon content influence the machinability of alloyed steels?

7. How will increased yield strength influence chip formation?

8. Why are negative cutting geometries normally recommended for short chipping materials like gray cast iron?

9. Why are positive cutting geometries recommended for the machining of Inconel?

10. What is the difference between the physical condition and physical property of a material?

11. How does the thermal expansion coefficient and conductivity of a material relate to its machinability?

12. How does a materials tensile strength relate to hardness?

13. What properties does the addition of Nickel, Chromium and manganese have on alloyed steels?

14. Rank the austenetic, ferritic, martensitic and PH stainless steels in terms of their machinability.

15. What are refractory alloys? Name three.

16. Why is grade 390 and 392 aluminum difficult to machine?

17. What is the difference between a cast and wrought material?

18. Why is 7075-T6 aluminum easier to machine than Ti-6Al-4V titanium?

TOOL SELECTION GUIDE BY GEOMETRY

Material	Preferred Geometry Axial	Preferred Geometry Radial	Recommended Cutter Series Drilling	Recommended Cutter Series Milling	Recommended Cutter Series Turning
Cast Iron Group	Neg. Neg.	Neg. Pos.	All Se Drill-Fix HTS & HTS-C	101,102 220,221 305*,306* 311*,313* All end mills All arbor mills	All ANSI M Style,1.33, 1.77,1.78, 1.70&1.38
Steel Group (Alloyed, Non-alloyed & Tool)	Neg. Pos. Pos.	Pos. Neg. Pos.	All Se Drill-Fix HTS & HTS-C	101,102 220,221 305*,306* 311*,313* 432,551,556 All end mills All arbor mills	All ANSI M & C Style 1.77,1.78, 1.70,1.38, & 1.33
Steel Group (Stainless)	Pos. Pos.	Pos. Neg.	Drill-Fix HTS & HTS-C	551,556 311*,313*	All ANSI C Style 1.77,1.78 1.70,1.38, & 1.33
Aluminum Group	Pos.	Pos.	Drill-Fix HTS & HTS-C	311,312,313 All end mills All arbor mills	All ANSI C Style 1.108 & 1.109
Copper Group	Pos.	Pos.	Drill-Fix HTS & HTS-C	311,312,313 551,556 All end mills All arbor mills	All ANSI C Style 1.108, 1.109 & 1.38
Nickel Group	Pos. Pos.	Pos. Neg.	Drill-Fix HTS & HTS-C	311*,313* 551,556 432 End & arbor mills with KMF inserts	All ANSI C Style** 1.38,1.70, 1.77,1.78, 1.33

TOOL SELECTION GUIDE BY GEOMETRY CONTINUED

Material	Preferred Geometry Axial	Preferred Geometry Radial	Recommended Cutter Series Drilling	Recommended Cutter Series Milling	Recommended Cutter Series Turning
Titanium Group	Pos. Pos.	Pos. Neg.	Drill-Fix HTS	311*,313* 551,556 432 End & arbor mills with KMF inserts	All ANSI C Style 1.38,1.70, 1.77 & 1.78
Refractory Group	Pos.	Pos.	Drill-Fix	311*,313* 551,556 432 End & arbor mills with KMF inserts	All ANSI C Style** 1.38,1.70, 1.77 & 1.78

* Only recommended when a lead angle is required.
** M style ANSI holder applies when using ceramic or whisker ceramic inserts.

Note : Geometry recommendations are based on milling cutter rake angles and are listed in order of preference. For example, double positive geometry is generally the best choice for the **Nickel Group,** followed by pos/neg geometry.

TRAINING MANUAL

DRILLING

INTRODUCTION

Drilling accounts for nearly 23% of world tool consumption for metal cutting operations. It ranks second to turning in its popularity as a metal cutting machining method. Drilling is the process used to create a cylindrical hole in a solid section of workpiece material. When using mechanical methods and tools, drilling involves rotational and linear feed motions.

The predominant drilling tool of the early 1800s was the single-point drill made of tool steel. This tool was capable of reaching cutting speeds of 16 surface feet per minute (SFPM). In the year 1820, the twist drill was invented. This invention provided improved chip evacuation and feed rates, but the actual cutting speed remained the same due to the tool material. In 1900, twist drills were introduced in high speed steel which raised the level of cutting speed to a range of 65-130 SFPM. The HSS twist drill was used for many decades with little further development in cutting speed or feed. The use of carbide tipped drills didn't occur until the year 1930 and cutting speeds were then elevated to 100-200 SFPM.

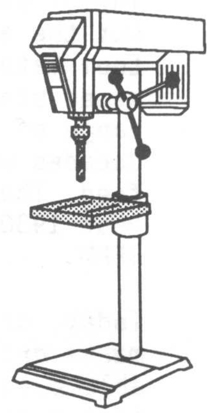
Today, drilling utilizes solid carbide coated high performance drills with special flute and point geometry, indexable carbide insert short hole drills and indexable carbide insert modular deep hole drilling systems. In addition, the laser, waterjet and electronic discharge machining (EDM) has replaced mechanical drills on more complex workpieces requiring holes. Our discussions on drilling, however, will be centered exclusively around solid carbide, indexable carbide and indexable carbide deep hole modular drilling systems.

DRILLING MACHINES

A wide variety of machine tools continue to be used to produce holes in workpieces using rotational and linear feed motions. Drilling is performed on machinery which has either horizontal or vertical spindles. Horizontal machinery is primarily used for drilling deep holes, while vertical machines are better suited to short holes, since gravity has less of an effect on chip evacuation on these machines. Lathes, drill presses and CNC machining centers are all used to produce holes in workpieces using drills. The following pictures and descriptions illustrate the differences between these machines:

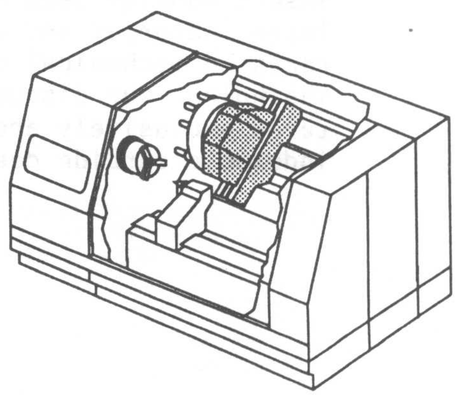
DRILL PRESS

The drill press is a column type vertical machine which is still used in tool rooms and on some production applications. High speed steel twist drills are often utilized on this type of equipment, since it often lacks the necessary rigidity required to support carbide tooling.



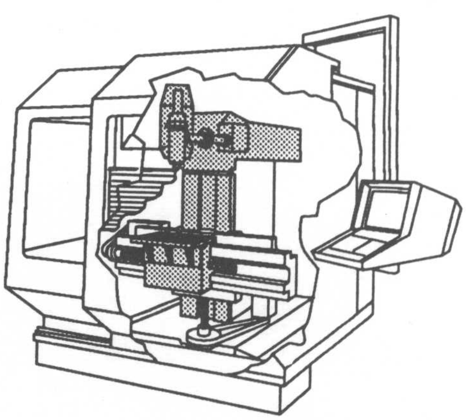
LATHE

The horizontal lathe is often used as a dedicated machine tool for deep hole drilling applications. Spade drills, gun drills, indexable carbide insert short and deep hole drills can be utilized on this type or machinery.



CNC MACHINING CENTERS

The evolution and prevalent use of high priced CNC machine tools has driven metal component manufacturers to search for high performance hole-making products to reduce cycle times. Today, many of these machines are designed around the use of solid carbide or coated indexable carbide insert drills. Therefore, their torque and horsepower is maximized at carbide cutting speeds.

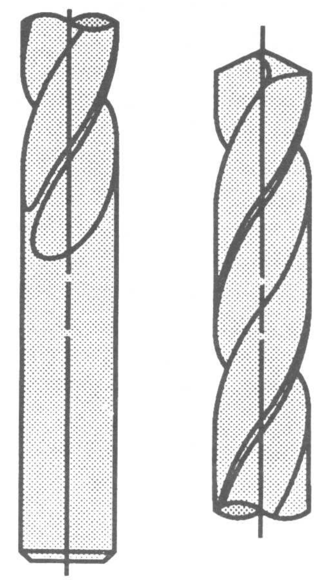


DRILLING TOOLS

Drilling tools have evolved from low cutting speed short hole tools with a single cutting edge to high performance indexable coated carbide tools with multiple cutting edges. The category of drilling tools includes HSS twist, spade, solid carbide, indexable insert, and modular deep hole drills as well as spot facing, counterboring, multi-step boring and trepanning tools. Let's discuss the features of each of these tools:

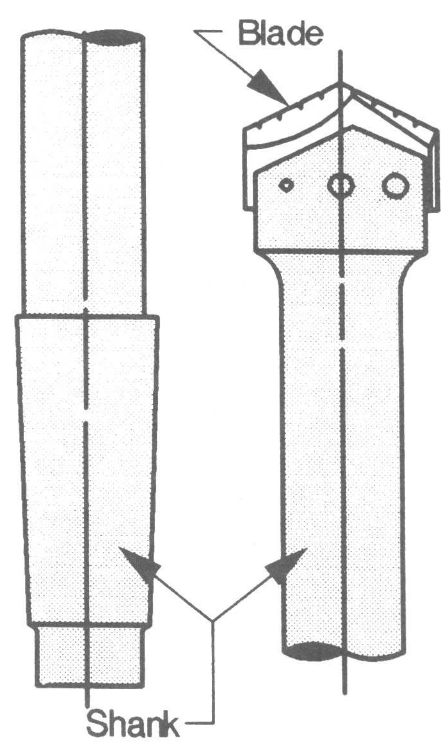
HSS TWIST DRILL

The high speed steel twist drill is a general purpose roughing tool used to produce holes from .004"- 4" in diameter. This style of drill is normally capable of attaining hole depths in the range of five times their drill diameter. High speed steel drills used to drill deeper holes either have special flute configurations or are retracted periodically during the drilling process to remove chips. The point geometry of a common HSS twist drill is a 118 degree included angle with a web thickness of 18-30% of the drill diameter.



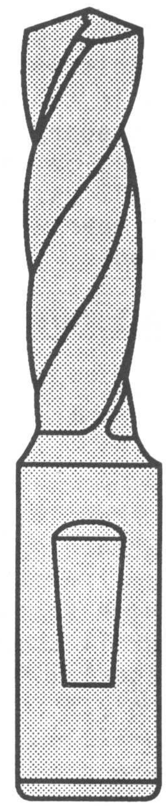
SPADE DRILL

Spade drills are generally used for deep hole drilling applications and large hole diameters (usually greater than 1.5"). These tools are made of two primary components, a steel shank and a replaceable drilling blade. Spade drill blades are made of carbide or, in some cases, HSS with a 130 degree drill point. This tool produces a great deal of torque, since they are often used at moderate cutting speeds and they have a long continuous cutting edge. There are indexable product offerings in the marketplace which are displacing the spade drill and its use is diminishing; however, when limited horsepower is available, it is often the best choice for drilling deep horizontal holes on lathes.



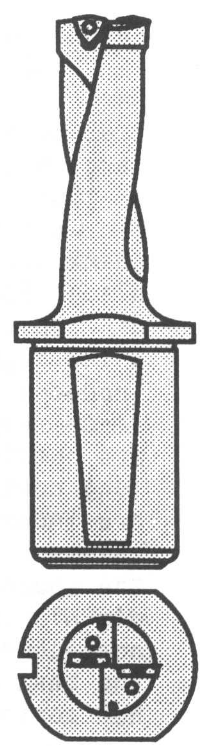
SOLID CARBIDE DRILLS

Solid carbide drills were designed as high performance replacement tools for the HSS twist drill. These tools operate at higher cutting speeds and feed rates with longer tool life than their HSS counterparts. The increased performance is a result of the tool geometry and the greater wear resistance and hot hardness of carbide. The Hertel solid carbide drill, known as the SE (sculptured edge) Drill, is designed to take advantage of carbide's unique physical properties. The SE drill is available from 3mm-20mm in diameter with the capability of drilling holes 5 to 7 times its diameter in depth. The web thickness of the SE drill is 30% of the cutting diameter, which provides excellent stability of the tool when combined with the sculptured cutting edge and 140 degree point. These tools are designed to provide maximum productivity and efficiency on NC machining centers.



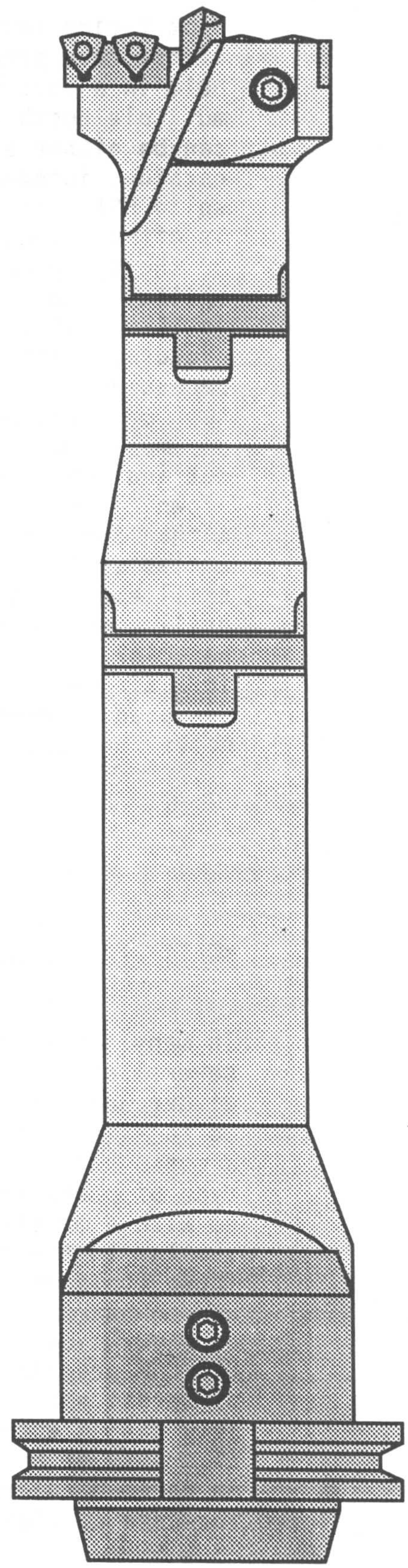
INDEXABLE INSERT DRILLS

All of the major cutting tool manufacturers offer an indexable insert drill. This type of tool is generally comprised of a steel body with insert pockets and machined flutes and multiple cutting edge indexable inserts with molded or pressed-in chip breakers. Hertel indexable insert drills (Drill-Fix) are capable of drilling two and a half times their diameter in hole depth at coated carbide cutting speeds of up to 1400 SFPM. These tools are manufactured in 16mm-82mm cutting diameters, normally with two indexable trigon inserts. To obtain optimum tool performance, the drill must be utilized on a machine with sufficient power and rigidity. This would include NC lathes, machining centers, and horizontal boring mills.



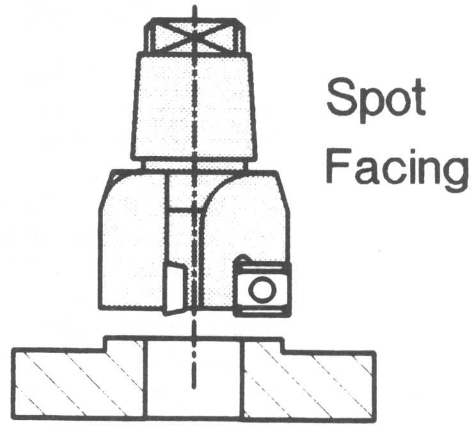
MODULAR DEEP HOLE DRILLS

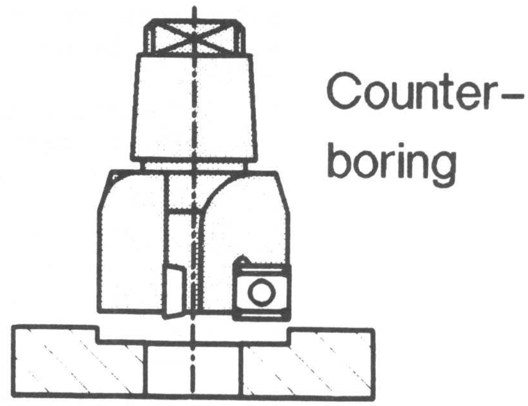
Modular drilling systems are becoming more popular, especially for deep hole applications. This concept becomes very practical at hole depths of three diameters, which is the point where indexable insert drills start to become unstable. Hertel has designed and introduced the HTS drilling system, which utilizes a HSS pilot drill to stabilize the tool as it enters the cut. The HTS modular drilling system has multiple components (drill heads, basic shanks, extensions, reducers and adaptors) which can be assembled in a variety of ways to produce hole depths of eight drill diameters or greater. This system operates at lower cutting speeds than a normal indexable drill, due to the limited speed capability of the pilot drill and the long overhang often associated with this type of tool. HTS is best suited for large boring mills and lathes. The HTS drill is a direct substitute for spade drills and is available from 1.77"- 6.69" in diameter. The HTS system uses replaceable cartridges to mount its trigon inserts, while the HTS-C product line has a fixed insert pocket. The HTS-C drilling system was designed for machining centers and NC lathes and parts with holes in the .79"- 1.77" diameter range. The standard HTS-C shanks will accommodate drilled hole depths of either three or five diameters. This system also utilizes a modular concept, since up to six different heads can be used on a single drill shank. The HTS-C system has a pilot drill for additional tool stability.



SPOT FACING AND COUNTERBORING

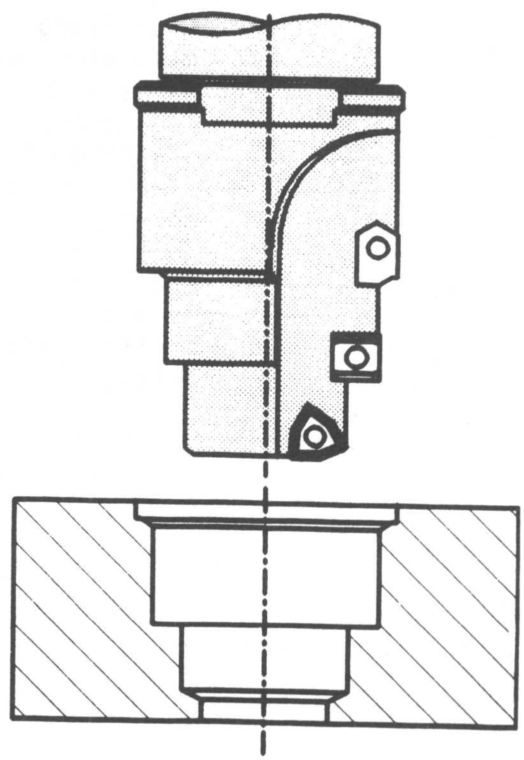
Spot facing involves the machining of the face perpendicular to the hole in a part boss, using a rotating multiple tooth cutter. These tools can be either solid carbide, brazed carbide, indexable carbide insert or solid HSS. The selection of the tool is often a function of the rigidity of the setup and the machine capability. The process of spot facing is illustrated in the top right hand picture. Counterboring involves the recessing of the area around a drilled, cast or cored hole. This is also accomplished using a tool with multiple cutting edges. A counterbored hole is shown to the right. The tool used for either operation is designed to cut a flat surface. The Hertel BF series drill employs the geometry of an SE drill combined with an indexable insert for the chamfering of holes all in a single tool. These tools are designed to prepare blind and through holes for tapping.





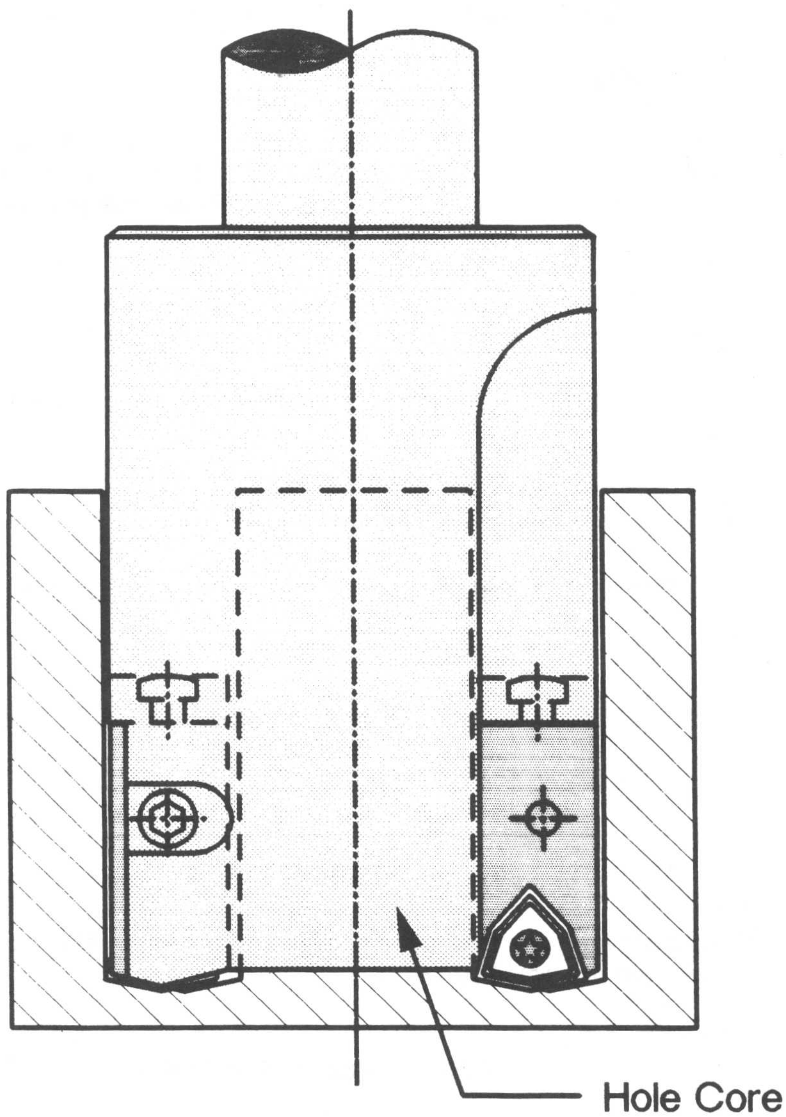
MULTIPLE STEP BORING

In many instances, a part with several concentric hole diameters either needs to be semi-finish drilled to size from solid or sized from a cored hole in one operation. This type of operation is easily completed with a multi-step combination tool similar to the picture at the right. These tools can drill, chamfer bore and countersink, all in a single tool configuration. Hertel has devised a CAD/CAM program which allows six different operations to be designed into a single combination tool with a variety of shank options as well as various tolerance specifications for each cutting diameter.



TREPANNING

This operation involves drilling a through hole by removing a portion of the material at the outer diameter and producing a core at the center of the hole. The core either drops out of the hole as the tool exits the part, or the core is retained inside the hollow body of the tool and removed from the tool after the hole is completed. The advantage of trepanning is to produce large diameter holes on lower horsepower machines. This is accomplished because horsepower in metal cutting is a function of the rate of metal removal. In trepanning, less than half of the metal in the hole is cut into chips. Therefore, in many cases the horsepower required to produce the hole is approximately 50% or less than if it were drilled with conventional tooling, which converts all of the material in the hole to chips.



CALCULATING CUTTING SPEEDS AND FEEDS

The drilling operation is a combination of linear (machine) and rotational (tool) movements. The rate in IPM (inches per minute) that a drill penetrates the workpiece is referred to as the machine feed. The speed in SFPM (surface feet/minute) that the periphery of the drill rotates is known as the cutting or surface speed. These two specific criteria are selected to either maximize tool life and productivity or to balance them.

The cutting speed in drilling is defined as the distance in feet traveled by a point on the periphery of the drill in one minute. The formula normally used to calculate cutting speed is as follows:

SFPM = (Drill Circumference) x (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:

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 x d x RPM

This formula can be used to determine the cutting speed at the periphery of any rotating drill. For a more in-depth explanation of the cutting speed, refer to the **"HERTEL TRAINING MANUAL FOR MILLING"**.

EXAMPLE #1
What cutting speed would you be running if you drilled a .25" diameter hole in cast iron at 5008 RPM?

$$SFPM = .262 \times d \times RPM = .262 \times .25 \times 5008$$

ANSWER : SFPM = 328

EXAMPLE #2
What RPM would you select to maintain a cutting speed of 394 SFPM on a 1015 low carbon steel part, if the drill you are using has a .75" diameter?

$$RPM = \frac{SFPM}{.262 \times d} = \frac{394}{.262 \times .75} = 2005$$

ANSWER : RPM = 2005

Once the cutting speed is 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. The application of solid carbide and modular deep hole indexable drills frequently involves determining a feed rate based on an advance per revolution which yields reliable chip control with little regard for the horsepower required. Indexable drill feed rates, however, are often a balance between the available machine horsepower and a machine feed which will produce reasonable 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. To determine the feed in IPM, use the following formula:

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

Where :

IPM = inches per minute

IPR = inches per revolution

RPM = revolutions per minute

Let's review some examples:

EXAMPLE #3
If you wanted to maintain a .017 IPR feed on the .75" drill used in EXAMPLE #2, what would this represent in feed rate in IPM?

$$IPM = IPR \times RPM = .017 \times 2005 = 34.1$$

ANSWER : IPM = 34.1

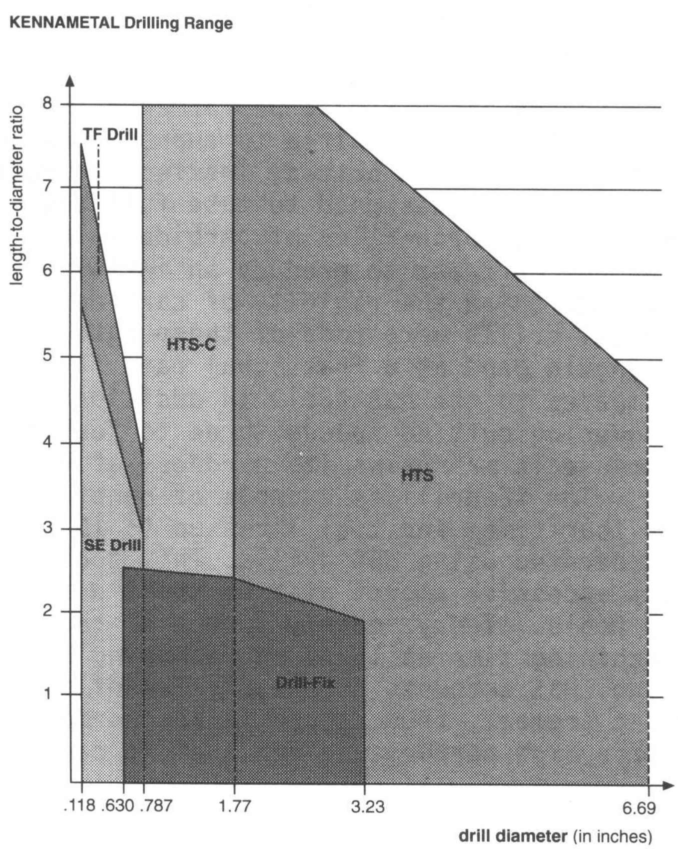
The selection of the feed per revolution (IPR) for any drill is done, using an application table often found in the catalog for that particular product series. The three examples shown use recommendations from the Hertel SE drill catalog for cutting speed (SFPM) and feed (IPR). The midpoint of the recommended feed range should always be selected as a starting point when applying a drill to test the stability of the part, machine, tool and fixturing. Once the stability of the machining system is established, the feed can be increased. The chart below is taken from the SE drill catalog, the operating values selected for EXAMPLES #1-3 were based on a .787" (20mm) diameter drill since this is the nearest size to the .75" SE drill indicated in each sample problem. Notice the feed used in EXAMPLE #3 is midway between the recommendation (.010-.024):

operating parameters for the application of SE drills

material group		tensile strength (psi)	BHN	material examples	cutting speed SFM	feed in inch per revolution for drill diamater				
						.197 in.- 5mm	.315 in.- 8mm	.472 in.- 12mm	.630 in.- 16mm	.787 in.-20mm
unalloyed steels	(C<.20%)	72,000	150	1015	394	.004-.007	.005-.012	.008-.016	.010-.020	.010-.024
unalloyed steels	(C.20%-.30%)	86,000	170	1025	361	.005-.008	.005-.012	.008-.016	.010-.020	.010-.024
unalloyed steels	(C.30%-.40%)	100,000	200	1030	328	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
alloyed steels	(C.30%-.40%)	94,000	185	4140	328	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
unalloyed steels	(C.40%-.50%)	115,000	230	1045	295	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
alloyed steels	(C.40%-.50%)	100,000	200	52100	295	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
alloyed steels		115,000	230	6150	263	.004-.007	.005-.010	.006-.014	.008-.018	.010-.020
alloyed steels		130,000	260	8620	230	.004-.007	.005-.010	.006-.014	.008-.016	.010-.018
alloyed steels		<144,000	290	H13	197	.003-.006	.005-.008	.005-.010	.006-.014	.008-.016
alloyed steels		>144,000	290	D3, D7	131-197	.003-.006	.005-.008	.005-.010	.006-.012	.008-.014
cast irons			150-200	G25	328	.008-.020	.012-.024	.016-.028	.016-.032	.016-.035
cast irons			200-220	G30	263	.008-.018	.012-.020	.016-.024	.016-.028	.016-.032
cast irons			220-250	G35	230	.006-.016	.010-.018	.012-.020	.012-.024	.012-.028
cast irons			250-320	G35	197	.005-.012	.008-.016	.010-.016	.012-.020	.012-.024

THE KENNAMETAL DRILLING PRODUCT SCOPE

Kennametal offers standard drilling products from .118"-6.69" in diameter with the ability to drill holes in a variety of length to diameter ratios. The SE drill is capable of drilling 6-7 diameters deep, while Drill-Fix can attain hole depths of 2.5 diameters deep, HTS-C has the ability to reach hole depths of 5 diameters and HTS has reached hole depths of 8 diameters. The graph bellow highlights the scope of the standard drilling product offered by Kennametal:



The "Drilling Depth To Diameter Ratio" shown on the graph above is determined by dividing the maximum attainable hole depth for an individual drill by its diameter. For example, a two inch diameter drill with the capability to produce a hole eight inches deep would have a depth to diameter ratio of four.

Depth To Diameter Ration = (Maximum Hold Depth)/(Drill Dia.)

THE SE SOLID CARBIDE DRILL

BACKGROUND

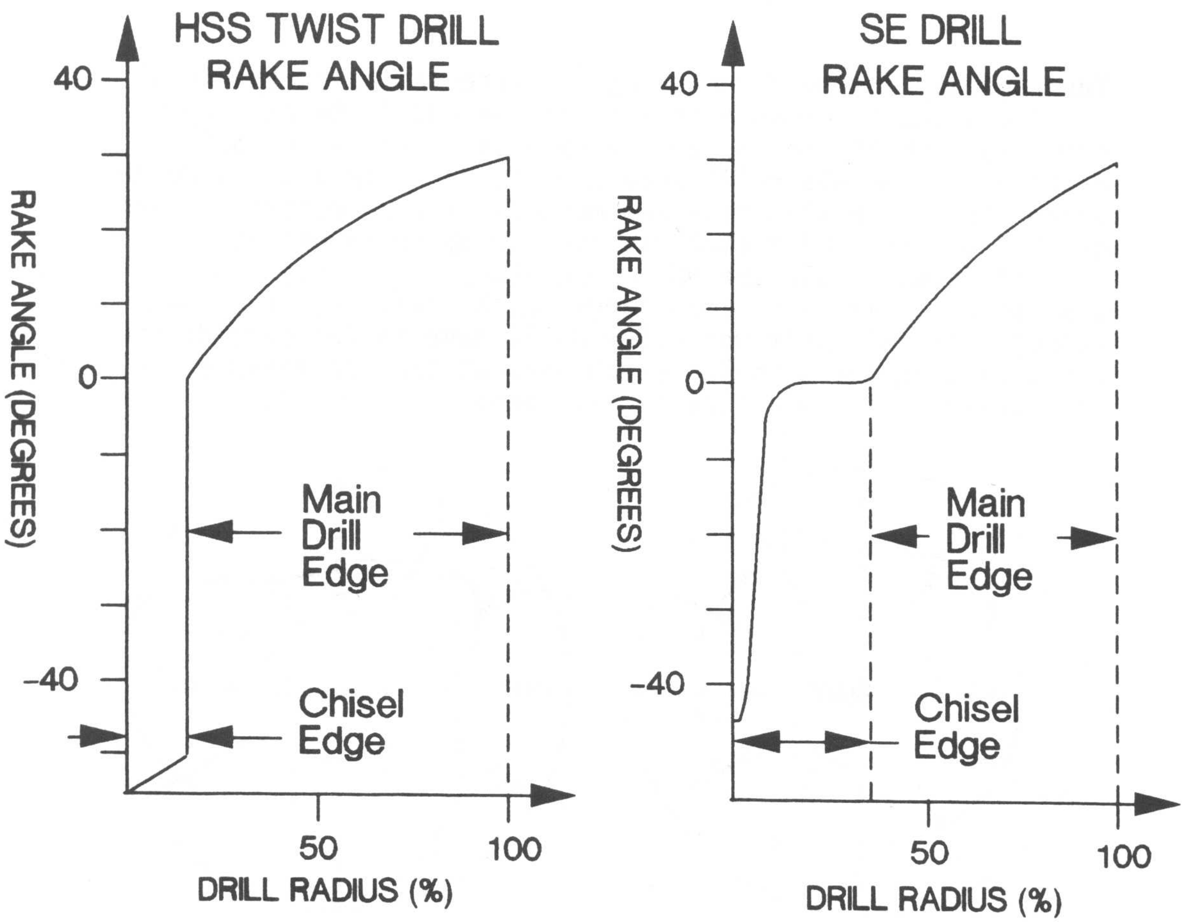
The SE (sculptured edge) high performance coated solid carbide drill was developed to displace the HSS (high speed steel) twist drill on shallow hole applications. This would include bolt hole mounting patterns as well as tapped holes. Many carbide producers have developed drills to displace HSS, but they often approached the design of these replacement tools by directly (in carbide) copying the dimensional size and geometry of existing HSS tools. This practice has led to mediocrity in design and ultimate performance, since these tools lacked the features required to support carbide and, hence, they were often high priced, ineffective mutations. The Hertel SE drill was designed to take full advantage of the inherent physical properties of carbide in terms of stiffness and hot hardness to produce an infinitely superior tool. HSS has one third the rigidity of carbide; therefore, if two identical drills were made of these materials, the carbide tool would bend at a fractional rate for a specific load when compared to the HSS tool. In addition, carbide is capable of enduring cutting speeds three to four times higher than HSS, because it maintains its hardness at elevated temperatures. The SE drill is capable of penetration rates in excess of four times and tool lives up to 10 times more than those generated using HSS drills. This tool was designed to reduce the excessive amount of time spent drilling on high cost machine tools. Today, nearly 30-50% of the total available machining time utilized on machining centers is spent drilling. HSS accounts for nearly 72% of the tools used in this market segment, thus, there is an enormous commercial opportunity for high performance replacement carbide tooling.

PRODUCT FEATURES

The SE drill has several important product features which distinguish it from HSS twist drills and other common solid carbide drills. Let's examine these product features individually:

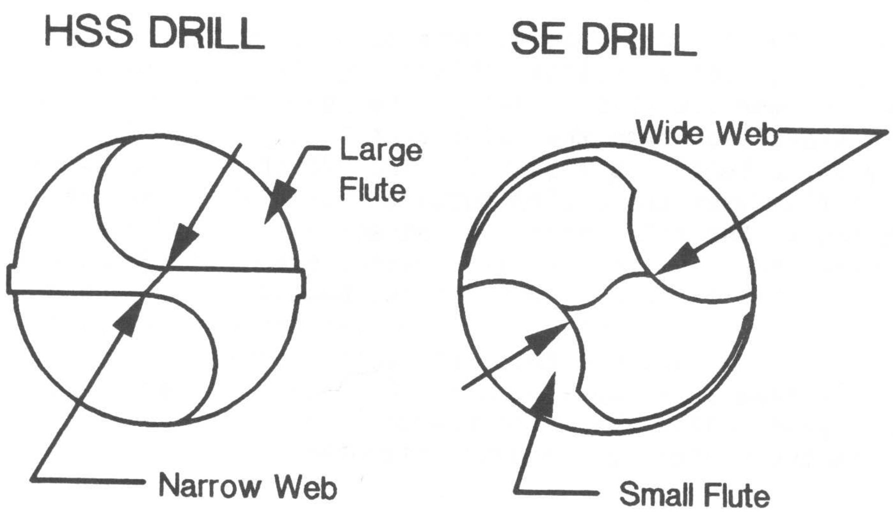
NEW POINT GEOMETRY

The sculptured edge drill point developed by Hertel has an extremely large active cutting chisel edge for high loadability and improved chip breaking. The point is ground with a smooth transition from the major cutting edge, thereby removing any potential stress peaks and providing a means of free chip flow over the entire cutting width. This feature contributes to the self-centering characteristics of the SE drill, since the tool is actively cutting metal from the center to the outer diameter. HSS drills, however, are ground with flat chisel points which rub and tear the work surface as they enter the cut, and thus, they will wander or walk if they aren't stabilized by a bushing or center-drilled pilot hole. The graph below shows the change in rake of the SE drill from the center to the outer diameter.

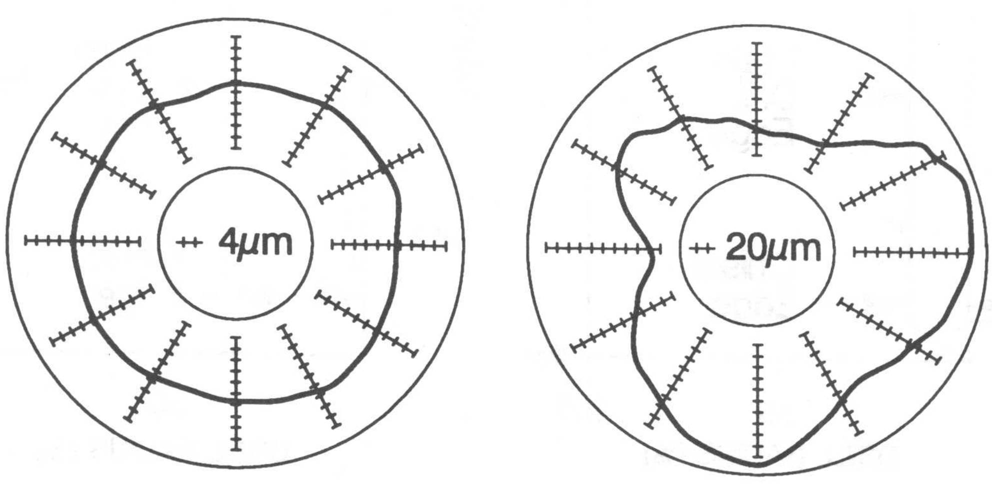


IMPROVED RIGIDITY

The combination of a smaller drill flute and an increased web thickness significantly enhances the stiffness of the SE drill, when compared to conventional carbide or HSS drills. This added rigidity resists the bending and torsional loads often encountered during the drilling process and, therefore, these tools are capable of greater penetration rates.



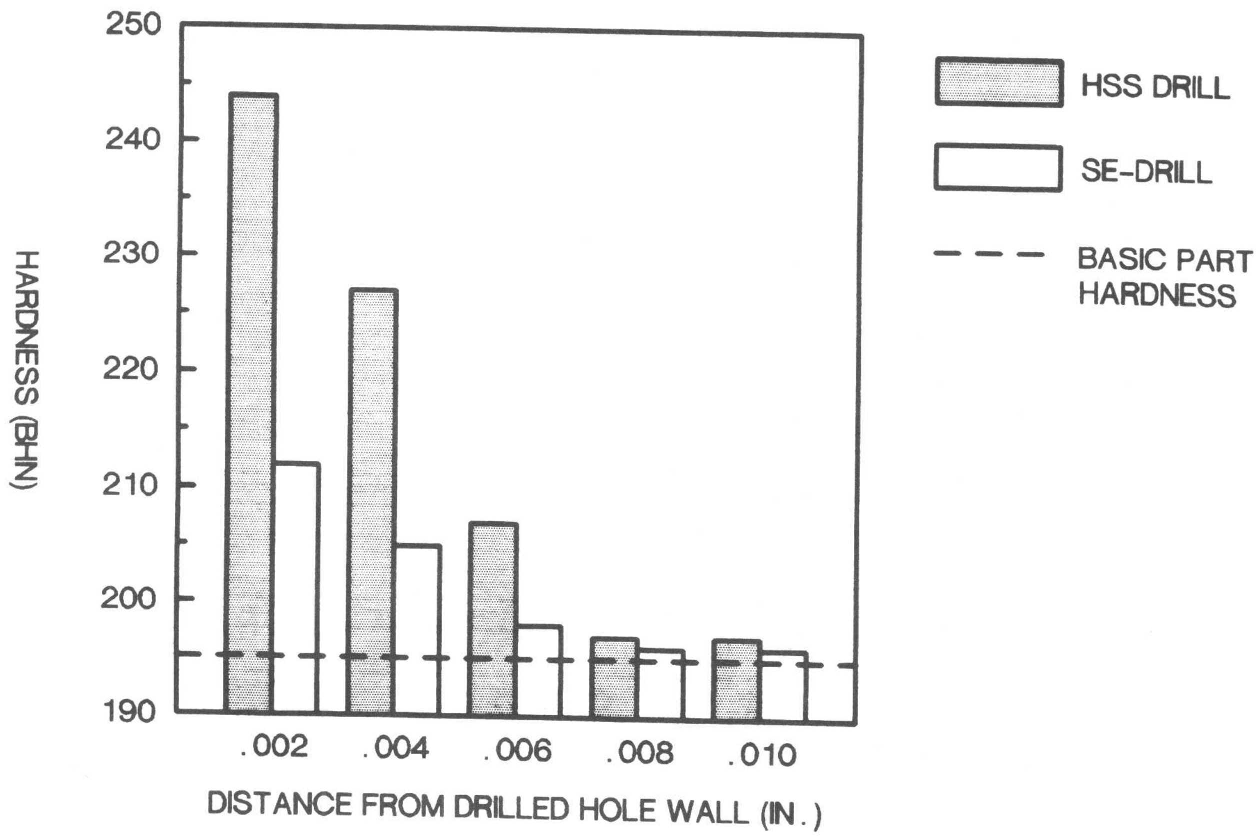
The rigidity and self centering characteristics of the SE drill combine to produce consistent concentric holes when compared with HSS tools. In the case shown below, a .63" diameter SE and HSS drill were used to drill in a steel work-piece. The SE drilled hole was measured for concentricity and graphed as shown with each division or space representing .00015" (4um), while the HSS graph shown at the right has a scale which is five times larger with divisions of .0008" (20um). The HSS hole variation would have fallen outside the boundaries of the circular graph without the increased scale. The variation in hole size and roundness is obvious.



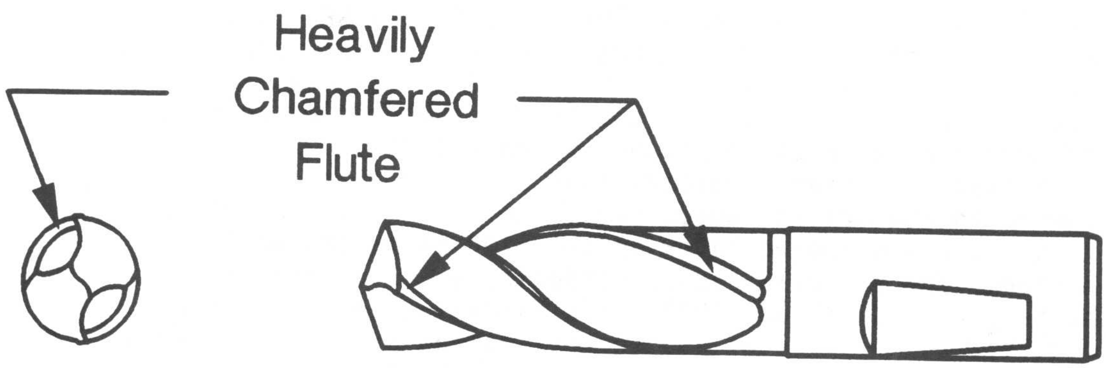
NEW FLUTE DESIGN

SE drills are designed with optimum flute and helix angle geometries, which provide excellent chip evacuation properties without sacrificing cross-sectional strength. The large heel chamfer on the trailing edge of the SE drill provides additional chip clearance to facilitate the flushing of chips with external coolant supplies, while the relatively small radius of curvature at the root of each SE drill flute aids in tightly coiling and breaking chips before they abrade the drilled hole sidewall. This particular feature significantly improves the surface quality of the drilled hole, and reduces the tendency to harden the hole surface. Tests conducted in Germany indicate that the poor chip evacuation common to HSS drills causes nearly 26% of the drill hole wall to develop a hardened layer up to .010" below the wall surface. The SE drill causes hardening on 7% of the wall surface at a depth of .004". This hardening condition can reduce the tool life of subsequent tapping operations. When tapping drill holes are produced by the SE drill, tap life has increased by nearly 50%. The graph below shows a comparison of HSS and SE drill hole wall hardening:

DRILLED HOLE WALL HARDENING

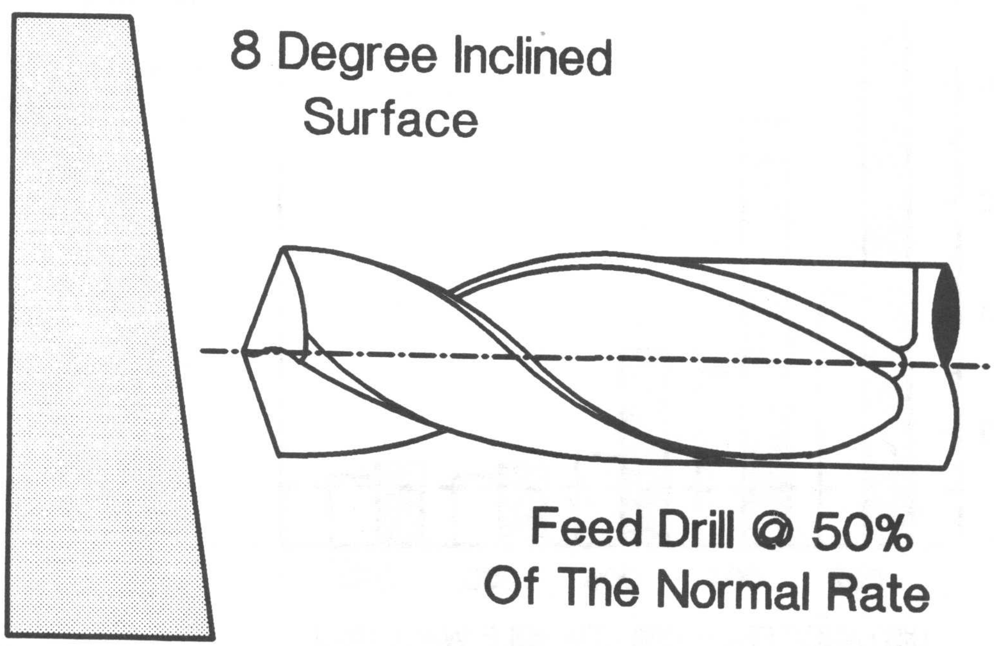


In some cases SE drills have eliminated the need for reaming due to their consistent size and their ability to produce finishes in the range of 32-63 RMS depending on the cutting conditions. Please refer to the pictures below which display the SE drill flute geometry:



STABILITY

Due to the SE drill's unique construction, the tool is capable of exceptional penetration rates without the need for bushings or center drilled holes for stabilization on entry. This allows the tool to enter angled workpiece surfaces of up to eight degrees after reducing the feed rate by 50%.



PRODUCTIVITY AND TOOL LIFE

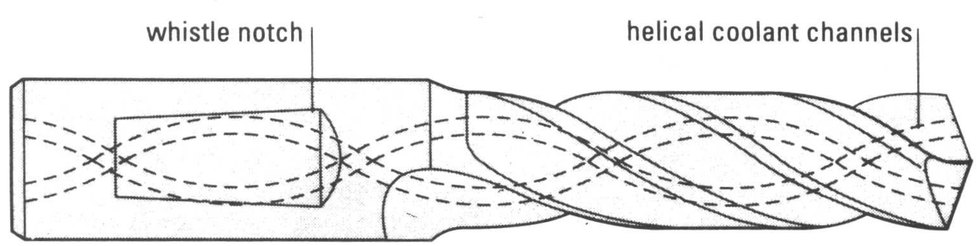
The SE drill utilizes a thin multi-layer CVD coating for improved heat and hence, wear resistance. This allows the SE drill to achieve cutting speeds four times greater than HSS. The SE drill is also capable of penetration rates 7.5 times those attained with HSS twist drills and 5 times greater than conventional carbide two-flute drills. The following example illustrates the productivity of the SE drill on structural steel when compared to other drills after two seconds of drilling:

Drill Style	Cutting Speed SFPM	Feed (IPR)	Hole Depth Reached
HSS	85	.006	.16"
Carbide 2 Flute	262	.003	.24"
Carbide 3 Flute	262	.006	.48"
SE	328	.013	1.26"

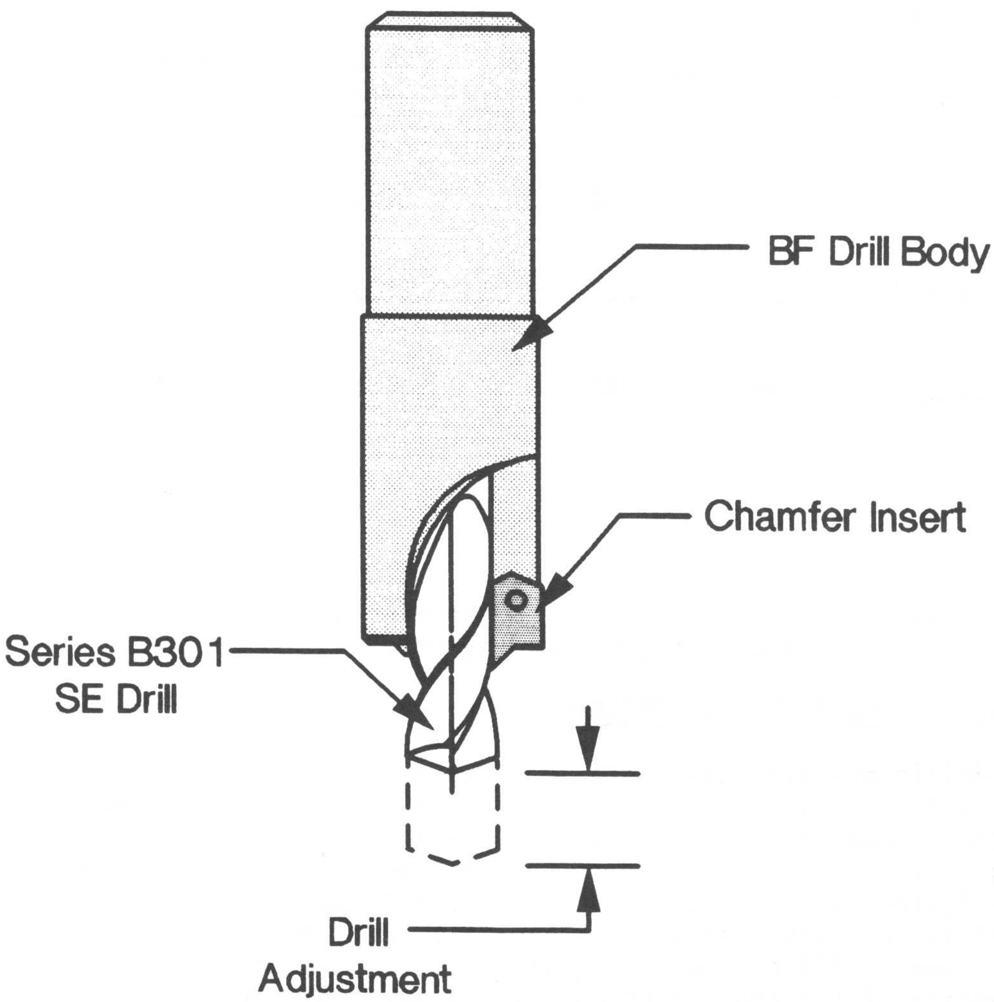
In addition, the SE drill has demonstrated tool life in excess of 10 times that generated by HSS.

COOLANT HOLE & CHAMFER OPTIONS

SE drills are also available in several sizes with coolant holes for drilling over three diameters in hole depth. The internal coolant holes improve chip evacuation by channeling fluid to the drill point. In addition, the tool is better lubricated and cooled, due to the coolant passing through the tool to the bottom of the drilled hole. A minimum operating pressure of 200 psi, with a fine filtering system to prevent clogging of the coolant holes, will provide the best tool performance.



The BF drill was developed after the SE drill for chamfering drill holes - in particular, tapped holes. This tool can be purchased with one or two chamfer inserts, depending on the drill diameter. The BF drill and chamfer inserts are screw mounted in a steel body, which has axial adjustment to precisely locate the drill relative to the chamfer insert. The range in axial adjustment of the BF drill is 1.2 times the drilling diameter. The chamfer inserts are available in three carbide grades and are easily changed out of the BF drill body with a torx screwdriver.



CONSISTENT CHIP CONTROL

The SE drill product line displays uniform chip control across the entire diameter range. The precise grinding and programed design of the SE drill guarantees uniform chip control over the entire product offering. The following pictures illustrate this product feature on four different materials:

SE Drill Diameter			
Material	5 mm	10 mm	18 mm
4147 STEEL			
6150 STEEL			
1015 STEEL			
4340 STEEL			

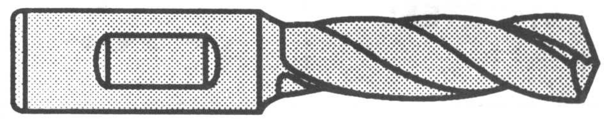
APPLICATION SCOPE

The SE drill is designed for rigid machine tools and not radial drills or manual drill presses. The point and flute geometry are especially suited for drilling carbon steel, alloy steel and ductile cast irons. The SE drill series B204W, B201F and B201V are all capable of drilling hole depths 3 diameters deep. The SE drill with coolant holes series B205F can drill in excess of 3 diameters, while B215 series with coolant hole can drill 6 to 7 times diameter. The BF drill with side chamfering inserts series B301S will produce hole depths 2.5-4.5 diameters deep, depending on which drill diameter is selected. None of the SE drills are recommended for most aluminums and titanium alloys. In some rare instances, the SE drill has been applied successfully on grades of aluminum when an adequate level of RPM was available. The SE drill (B210 series) can also be used on stainless steels.

PRODUCT SCOPE

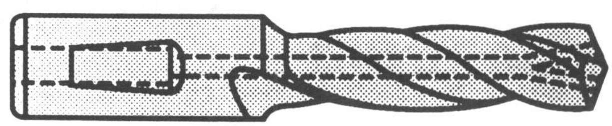
SE DRILL SOLID CARBIDE

The SE drill inch series B204W is available from .25"-.75" in diameter with inch shanks. There are 21 standard sizes available in this product series with Wheldon flats and inch diameter shanks. The metric series B201F/V SE drill is marketed from 3mm-20mm in diameter. These tools are produced with metric shanks and Whistle notches. There are 110 catalog standard drills in this product series of which 47 are stocked items.



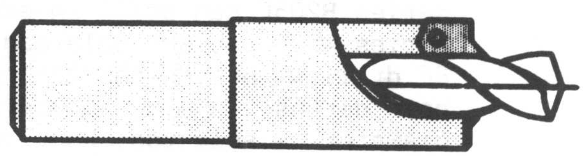
SE DRILL WITH COOLANT HOLES

The SE drill metric series B205F with coolant holes is available from 10.2mm-20mm in diameter. This product series has a straight shank with a Whistle notch. There are 24 std. SE drill sizes inventoried with this coolant hole option.



BF DRILL WITH CHAMFER STATION

The BF drill metric series B301S with SE type drill, steel drill body and indexable side chamfering insert is available from 3.4mm-18mm in diameter. The steel drill body, has an option for a straight shank or Whistle notch. There are 102 catalog standard BF chamfering tools.



OPERATING PARAMETERS

The proper selection of feeds and speeds is essential in the successful application of the SE drill. The SE drill catalog has a comprehensive listing of cutting speeds and feeds for a variety of common workpiece materials. There is one cutting speed (SFPM) listed for each specific material. The speeds should be selected based on the depth of hole to be drilled. For instance, the SE drill catalog specifies a 10% increase in cutting speed from the catalog value listed when the hole depth is one diameter deep. In those cases where the hole depth is two diameters deep, the cutting speed should equal the catalog specification. Hole depths exceeding 2.5 diameters should be drilled at 80% of their recommended cutting speed. The selection of cutting speeds above the values shown in the SE drill catalog will cause reductions in tool life.

A chart of feed rates is also shown in the SE drill catalog, based on the hole diameter to be drilled and the workpiece material. In each case, a range of feeds (inches/ revolution or IPR) is specified. Since the catalog guidelines for both speeds and feeds are designed to be used as a starting point when setting up a drilling operation, the feed rate should always be selected at the midpoint of the range specified. This type of approach will prevent initial tool breakage by allowing the SE drill to make a few trial holes to establish whether the machine setup, fixturing and workpiece is rigid at a moderate feed condition.

The following specifications are shown in the SE drill catalog for speeds and feeds:

CHART #1 SOLID DRILL

operating parameters for the application of SE drills

material group		tensile strength (psi)	BHN	material examples	cutting speed SFM	feed in inch per revolution for drill diamater .197 in.- 5mm	.315 in.- 8mm	.472 in.- 12mm	.630 in.- 16mm	.787 in.-20mm
unalloyed steels	(C<.20%)	72,000	150	1015	394	.004-.007	.005-.012	.008-.016	.010-.020	.010-.024
unalloyed steels	(C.20%-.30%)	86,000	170	1025	361	.005-.008	.005-.012	.008-.016	.010-.020	.010-.024
unalloyed steels	(C.30%-.40%)	100,000	200	1030	328	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
alloyed steels	(C.30%-.40%)	94,000	185	4140	328	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
unalloyed steels	(C.40%-.50%)	115,000	230	1045	295	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
alloyed steels	(C.40%-.50%)	100,000	200	52100	295	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
alloyed steels		115,000	230	6150	263	.004-.007	.005-.010	.006-.014	.008-.018	.010-.020
alloyed steels		130,000	260	8620	230	.004-.007	.005-.010	.006-.014	.008-.016	.010-.018
alloyed steels		<144,000	290	H13	197	.003-.006	.005-.008	.005-.010	.006-.014	.008-.016
alloyed steels		>144,000	290	D3, D7	131-197	.003-.006	.005-.008	.005-.010	.006-.012	.008-.014
cast irons			150-200	G25	328	.008-.020	.012-.024	.016-.028	.016-.032	.016-.035
cast irons			200-220	G30	263	.008-.018	.012-.020	.016-.024	.016-.028	.016-.032
cast irons			220-250	G35	230	.006-.016	.010-.018	.012-.020	.012-.024	.012-.028
cast irons			250-320	G35	197	.005-.012	.008-.016	.010-.016	.012-.020	.012-.024

CHART #2 COOLANT HOLE DRILL

operating parameters for the application of coolant hole SE drills

material group	strength hardness	material examples	cuttingspeed ft./min.	feed in inch per revolution for drill diamater .402 in.- 10.2mm	.472 in.- 12mm	.630 in.- 16 mm	.787 in.- 20 mm
carbon steel (C<0.2%)	73 Kpsi	1015,1018	400	.005-.012	.008-.016	.010-.020	.010-.024
carbon steel (C 0.2-0.3%)	87 Kpsi	1020	360	.005-.012	.008-.016	.010-.020	.010-.024
carbon steel (C 0.3-0.4%)	102 Kpsi	1035	330	.006-.014	.010-.018	.012-.024	.014-.024
alloy steel	94 Kpsi	4140	330	.006-.014	.010-.018	.012-.024	.014-.024
carbon steel (C 0.4-0.5%)	116 Kpsi	1045	300	.006-.014	.010-.018	.012-.024	.014-.024
alloy steel	102 Kpsi	4340	300	.006-.014	.010-.018	.012-.024	.014-.024
alloy steel	116 Kpsi	6150	260	.005-.010	.006-.014	.008-.018	.010-.020
alloy steel	130 Kpsi	P4	230	.005-.010	.006-.014	.008-.016	.010-.018
alloy steel	145 Kpsi	S1	200	.005-.008	.005-.010	.006-.014	.008-.016
alloy steel	> 145 Kpsi	D3	130-200	.005-.008	.005-.010	.006-.012	.008-.014
cast iron	150-200 BHN	—	330	.012-.024	.016-.028	.016-.032	.016-.035
cast iron	200-220 BHN	—	260	.012-.020	.016-.024	.016-.028	.016-.032
cast iron	220-250 BHN	—	230	.010-.018	.012-.020	.012-.024	.012-.028
cast iron	250-320 BHN	—	200	.008-.016	.010-.016	.012-.020	.012-.024

CHART #3 BF DRILL

operating parameters for the application of BF drills

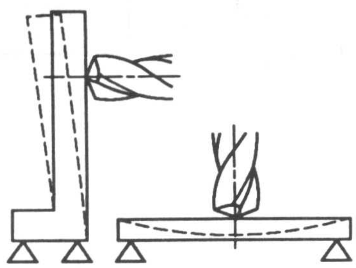
material group		tensile strength (psi)	BHN	material examples	cutting speed SFM	feed in inch per revolution for drill diamater .197 in.- 5mm	.315 in.- 8mm	.472 in.- 12mm	.630 in.- 16mm	.787 in.-20mm
unalloyed steels	(C<.20%)	72,000	150	1015	394	.004-.007	.005-.012	.008-.016	.010-.020	.010-.024
unalloyed steels	(C.20%-.30%)	86,000	170	1025	361	.005-.008	.005-.012	.008-.016	.010-.020	.010-.024
unalloyed steels	(C.30%-.40%)	100,000	200	1030	328	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
alloyed steels	(C.30%-.40%)	94,000	185	4140	328	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
unalloyed steels	(C.40%-.50%)	115,000	230	1045	295	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
alloyed steels	(C.40%-.50%)	100,000	200	52100	295	.005-.010	.006-.014	.010-.018	.012-.024	.008-.024
alloyed steels		115,000	230	6150	263	.004-.007	.005-.010	.006-.014	.008-.018	.010-.020
alloyed steels		130,000	260	8620	230	.004-.007	.005-.010	.006-.014	.008-.016	.010-.018
alloyed steels		<144,000	290	H13	197	.003-.006	.005-.008	.005-.010	.006-.014	.008-.016
alloyed steels		>144,000	290	D3, D7	131-197	.003-.006	.005-.008	.005-.010	.006-.012	.008-.014
cast irons			150-200	G25	328	.008-.020	.012-.024	.016-.028	.016-.032	.016-.035
cast irons			200-220	G30	263	.008-.018	.012-.020	.016-.024	.016-.028	.016-.032
cast irons			220-250	G35	230	.006-.016	.010-.018	.012-.020	.012-.024	.012-.028
cast irons			250-320	G35	197	.005-.012	.008-.016	.010-.016	.012-.020	.012-.024

TROUBLESHOOTING GUIDE

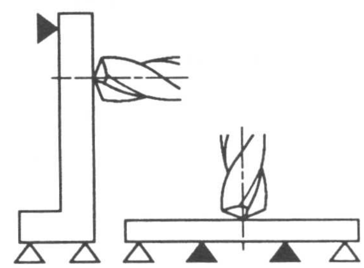
The SE Drill is a high performance tool requiring some attention to details in terms of the machine, part and fixturing in order to obtain maximum productivity. Please review the following recommendations which will help identify, remedy and therefore eliminate common problems encountered when using these tools:

INCORRECT **CORRECT**

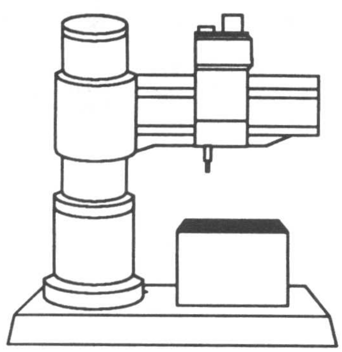
TOOL VIBRATION



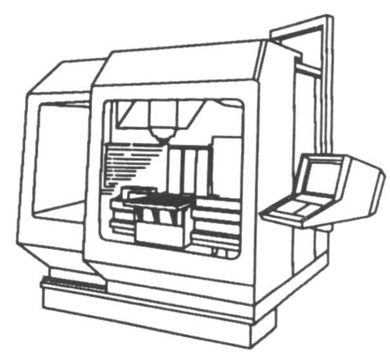
This condition is frequently caused by insufficient part rigidity or inadequate fixturing. Vibration will often cause retraction marks on the drilled hole wall or notches on the cutting edge. To eliminate this condition, provide additional support to the part or reduce feeds and speeds.



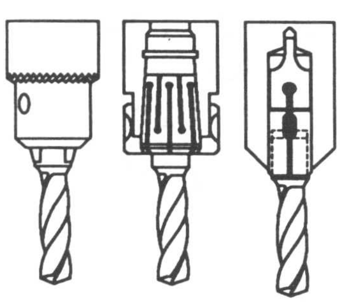
UNSTABLE MACHINERY



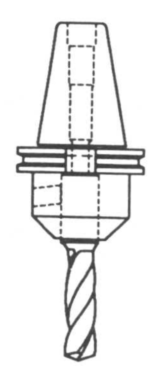
Machines which are unable to oppose the SE drill feed force without movement will include radial drills and multi-spindle drill heads. SE drills will exhibit wear on their guiding chamfers and generate retraction marks on the drilled hole wall if machine movement occurs. To rectify this condition, use machining centers, turning centers or NC lathes with sufficient rigidity and power to utilize the tool efficiently.



INADEQUATE TOOLHOLDER

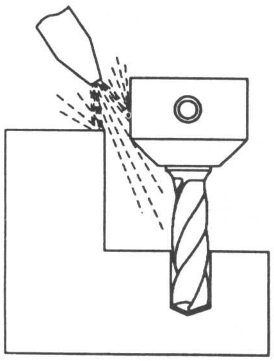


The SE drill requires precise stable toolholders with no more than .001" runnout and the ability to transmit torque without slippage. Collet holders should be considered only for drills under .5" in diameter, since the SE drill produces greater torque than HSS. Premature lateral wear and breakage will occur without the proper toolholder.

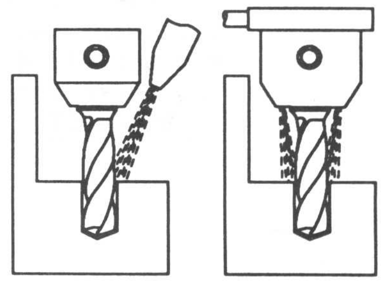


INCORRECT **CORRECT**

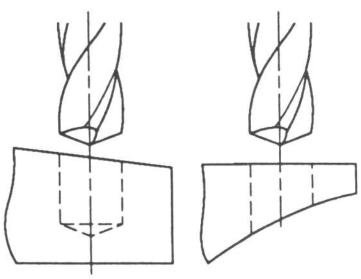
CHIP EVACUATION



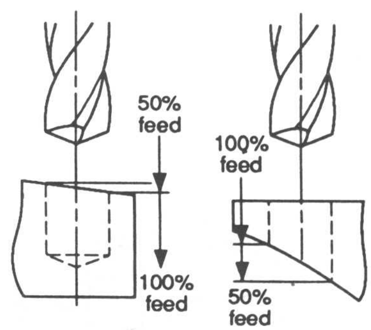
Improper chip evacuation can occur if the coolant nozzle is not properly directed on the periphery of the tool in line with the tool axis. This condition can lead to blue or brown chips, undersize holes, drill breakage or wear on the guiding chamfers of the drill. To optimize chip evacuation at least one coolant jet must be directed at a slight angle to the tool axis.



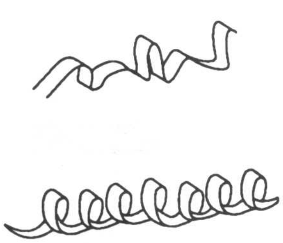
INCLINED SURFACES



Notching on the cutting and chisel edge, drill breakage and wear on the guiding drill chamfer are all symptoms of drilling inclined surfaces at normal SE feed rates. The SE drill is capable of drilling inclined surfaces of up to 8 degrees if the feed is reduced by 50%. If the angle exceeds eight degrees, pre-mill flats on the part surface. When drilling through an inclined surface, reduce feeds by 50% as the drill exits the part.



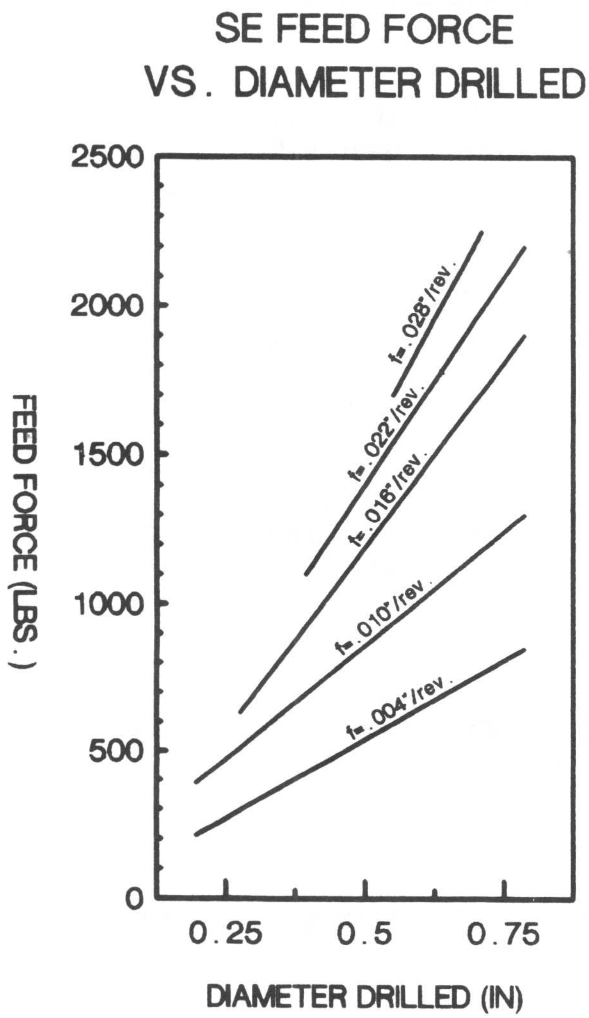
CHIP CONTROL

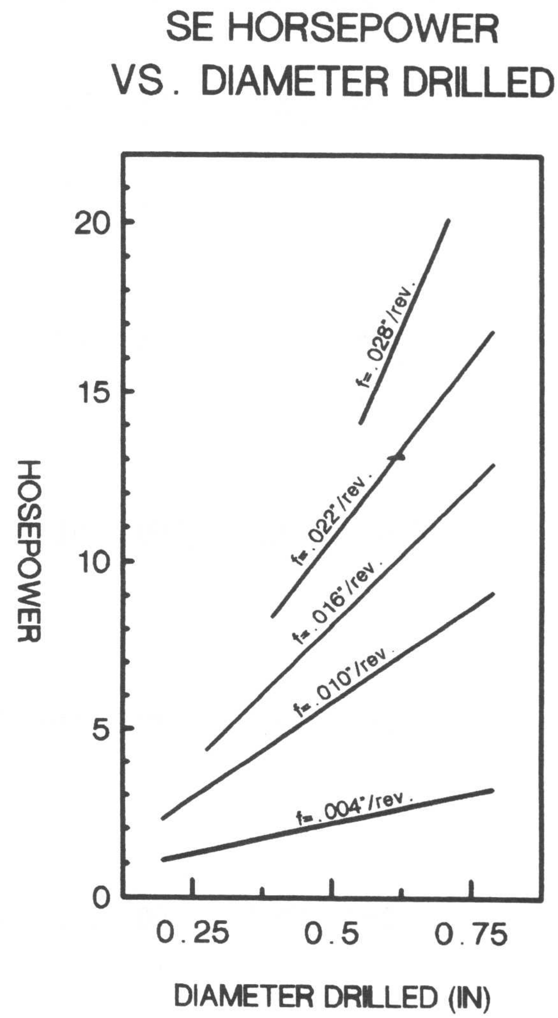


Poor chip control will cause SE drill breakage and rough hole finishes. When the SE drill is not producing short distinct helical chips, increase the feed rate or introduce an interrupted drill cycle which periodically retracts the drill .010" from the hole bottom and then continues drilling. If blue chips are evident, either increase the coolant supply or reduce the cutting speed.

FEED FORCE AND HORSEPOWER

The graphs shown below illustrate the feed force produced and power consumed when drilling a 116,000 psi high carbon steel at 300 SFPM. Five different feed rates are provided on each graph to allow selection of the appropriate cutting conditions in advance of actually running an SE drill on a low power machine tool. Therefore, if a question arises as to whether a prospective machine tool has adequate power or thrust capability to run an SE drill, the charts can be used to make that determination.





EXAMPLE #4

A customer has a seven horsepower machining center and would like to drill a 15mm hole at 300 SFPM and a .022" feed per revolution. Can he drill this hole?

ANSWER : According to the chart, at this feed rate he would need almost 12.5 horsepower. Therefore, either he needs a larger machine or he must utilize a lower feed rate (.010"-.014"/rev.).

FIELD TEST RESULTS

The following tests were completed using solid SE drills with external coolant supplies and have all led to purchase orders by each customer:

Drill Diameter	RPM	Cutting Speed SFPM	Feed Rate IPR	Feed Rate IPM	Hole Depth	Material	# of Holes
.500"	1700	223	.009	15.3	.5" Thru	1035 stl. gear blk.	2400
3.3mm	4000	136	.004	16.0	.2"	Titanium	120
6.8mm	4000	280	.015	60.0	.8"	Duct. Iron 180 BHN	1200
11.5mm	2000	237	.010	20.0	.25"	1010 stl. 30 RC	na
.375"	4000	390	.006	25.0	.50"	1020 stl.	1860
5.5mm	3500	230	.010	35.0	.45" Thru	4140 stl. 42 RC	850

THE DRILL-FIX INDEXABLE DRILL

BACKGROUND

Drilling holes from a solid commands a significant portion of the current hole-making marketplace. Drill-Fix was developed in 1970 to address the need for high performance drills over .5" in diameter, to accompany the prolific use of NC machinery, and to displace HSS twist drills on many of the existing shallow hole (2.5 diameters in depth) applications. The indexable drill can reduce machine cycle times by introducing higher penetration rates due to its efficient use of the available machine power. The geometry improvements common in indexable drills reduce cutting friction and provide the necessary chip control to accommodate high production drilling applications. The flexibility of indexable drills allow the end user to select the appropriate carbide grade, insert radius and chip breaker to optimize the process of drilling his specific workpiece material.

The Drill-Fix indexable drill utilizes positive rake trigon inserts which have virtually become the industry standard worldwide. These tools are significantly more versatile than the HSS twist drills they often replace. For example, on a NC lathe, a Drill-Fix can drill from solid, bore, face, chamfer and off-center drill larger hole diameters. In addition, these tools have demonstrated documented increases in drilling efficiency of 300-500%, when compared with HSS twist drills.

As mentioned earlier, 72% of the current drilling market is dominated by HSS tooling and, therefore, the potential to displace these tools with superior products like Drill-Fix is still enormous.

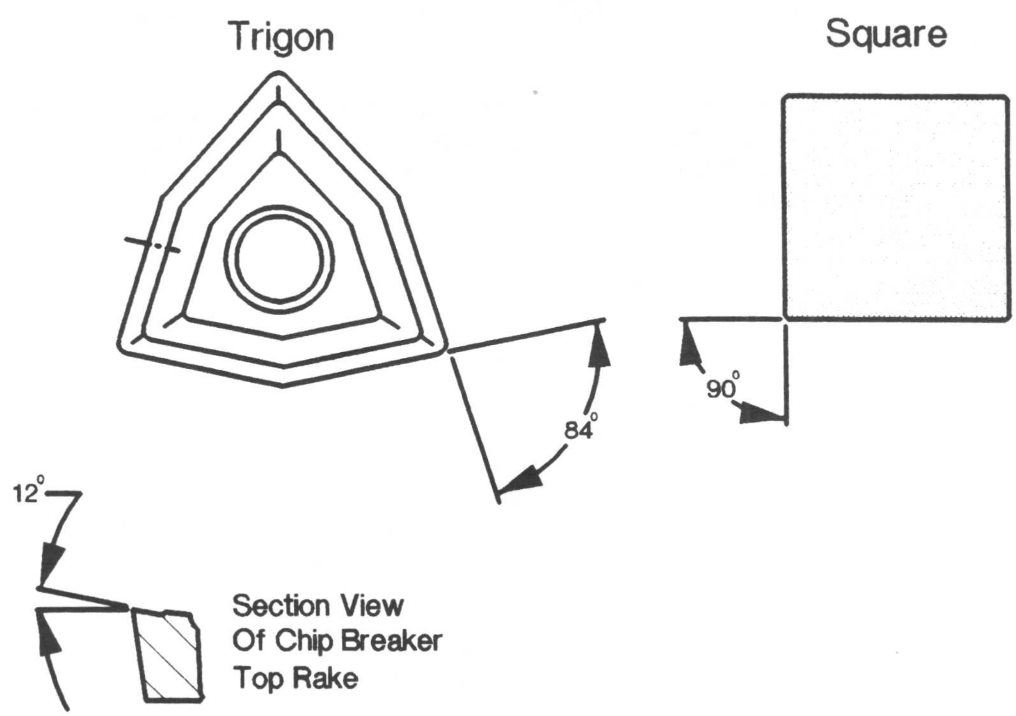
PRODUCT FEATURES

The Drill-Fix system is unique among indexable drills due to its attention to design details. The system is capable of drilling holes from a solid workpiece up to two and a half diameters in hole depth. Therefore, Drill-Fix is considered a short hole indexable drill. Let's examine the features of this unique product:

POSITIVE CUTTING ACTION AND STRENGTH

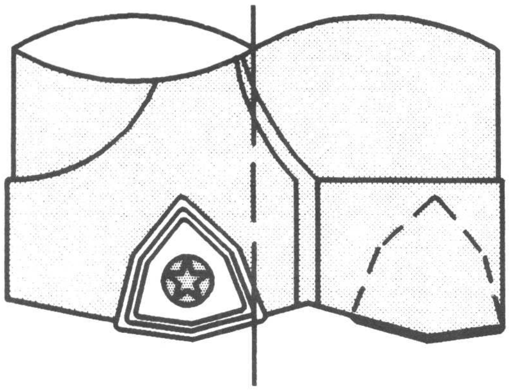
Drill-Fix utilizes positive rake trigon inserts mounted in a neutral attitude in the drill body. The molded chip breaker of the trigon insert has either a 12 or 24 degree positive rake (depending on the insert selected), which creates the positive effective cutting geometry common to the entire product line. Lower cutting pressure and reductions in the axial cutting force produce improved tool life when compared to negative rake indexable drills. The inserts are ground with an eight degree clearance angle, which provides adequate clearance to develop long wear lands for extended tool life.

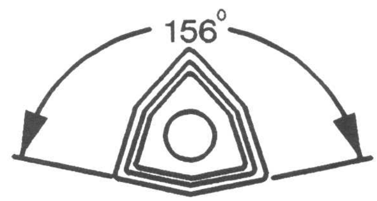
The 84 degree included corner angle of the trigon insert provides Drill-Fix with a cutting edge configuration that is virtually as strong as a square insert. This basic strength in shape is enhanced further by the option to select a .031" corner radius, if the standard .016" radius lacks the required strength under severe cutting conditions.



STABILITY

Drill-Fix is designed as a short hole drilling system (2.5 diameters in depth). Therefore, the overall length of the tool is kept to an absolute minimum. This provides the added rigidity necessary to drill on non-flat slopping or rounded surfaces without pre-centering. The large 156 degree included wedge angle on the adjacent sides of the trigon insert gently eases the tool into the cut and acts to stabilize the Drill-Fix once it's fully engaged.

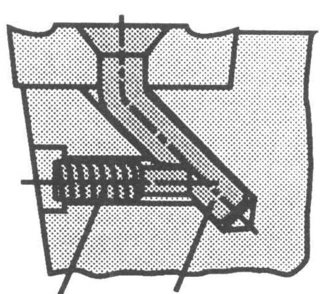


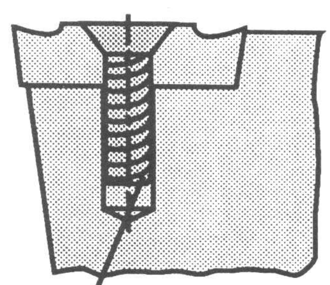


SECURE INSERT LOCKING METHOD

Drill-Fix drills benefit from the strength of the Fix-Perfect clamp stud clamping system in 1.18"(30mm)-3.23"(82mm) diameters. This system secures the insert in the direction of the main cutting force, provides unhampered chip flow, fast indexing and very few spare parts.

This system would be used in the smaller drill diameters if sufficient space was available. In the .63"(16mm)-1.18"(30mm) diameter drill range, torx screws are used to mount the trigon insert.



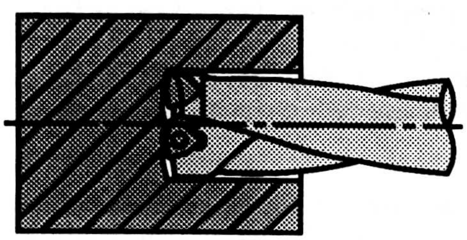


SUPERIOR FLUTE DESIGN & VERSATILITY

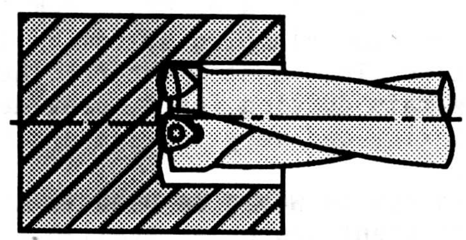
The Drill-Fix's unique helical flute design provides excellent chip evacuation along with an extremely rigid tool crossection, which opposes the radial forces common to an indexable drilling operation. The Hertel flute design incorporates a modest tapered helix angle, which efficiently evacuates large chips on both rotating (machining centers) and stationary (lathes) applications. The helix angle of each drill flute varies with the drill size, resulting in the stiffest possible body construction. This design approach assists in preventing deflection and vibration while improving tool life.

The Drill-Fix system is capable of drilling holes from solid, boring, facing, chamfering and off-center drilling holes larger than the nominal drill diameter. The .63"-3.23" diameter drilling range can be covered with only 18 drills by offsetting the drill in the direction of the out-board insert on a lathe (stationary application). See the Drill-Fix catalog for a complete chart of offset dimensions.

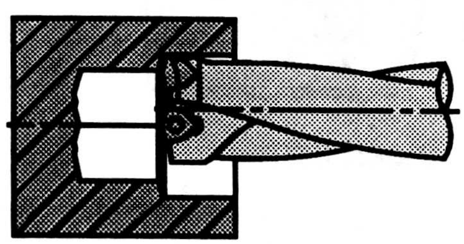
to diameter



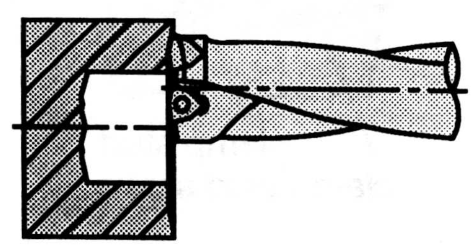
larger than diameter



boring

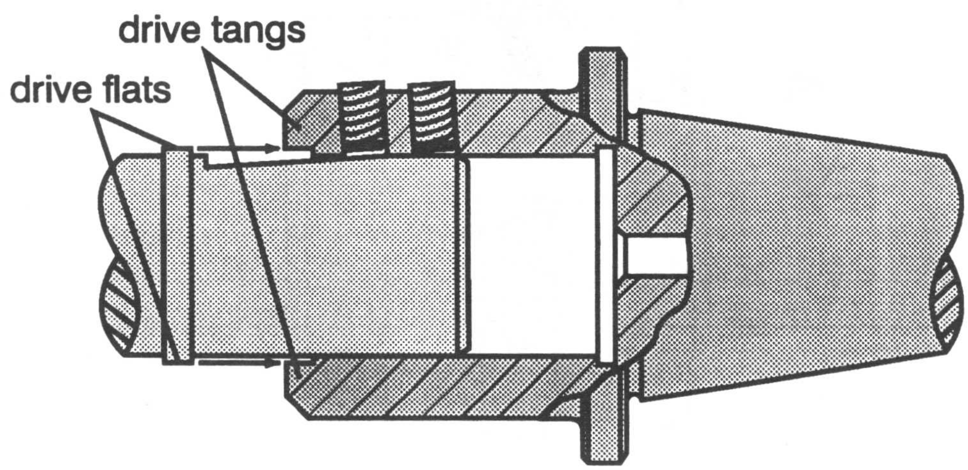


facing



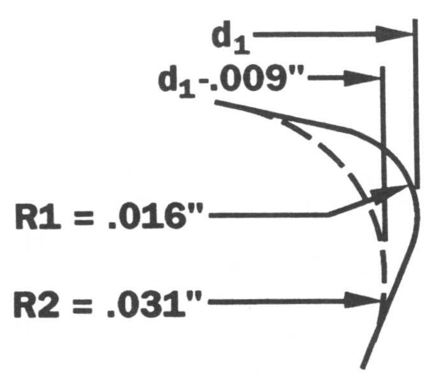
UNIQUE DRIVE SYSTEM

The Drill-Fix drill is driven via a two degree drive flat on the tool shank and two opposing flats on the drive flange. The flange flats locate into keyway type drive tangs on Drill-Fix style adaptors. This design is superior to the normal set screw drive system. All Drill-Fix shanks have coolant access holes radially from the side and axially from the back. The axial coolant hole is sized with a 1/4" NPT pipe thread.



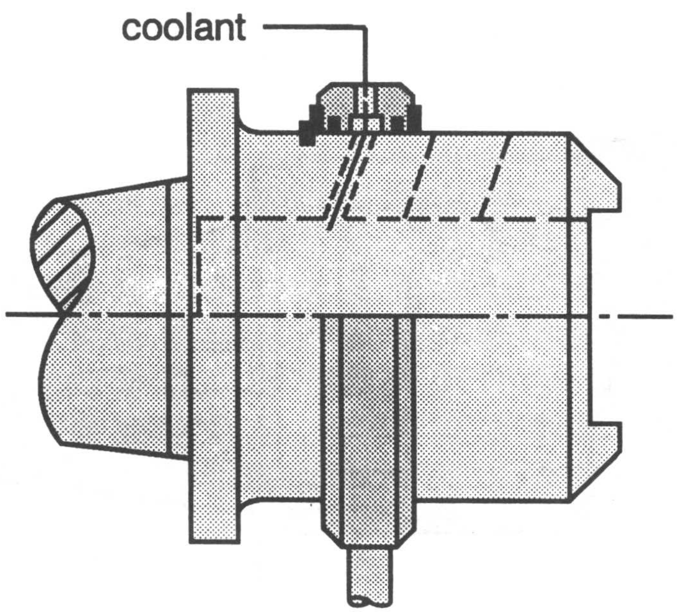
DIAMETER VARIATION

The diameter of a Drill-Fix drill can be altered either by offsetting the drill or by interchanging the .016" with a .031" radiused trigon insert. This reduces the drill diameter by .009". The normal tolerance on a Drill-Fix is the nominal diameter ± .008".



COOLANT ACCESS OPTIONS

Coolant is required to flush the chips from the drilled hole as Drill-Fix is fed through the workpiece. The coolant can be supplied to the drill in several different ways. On machining centers, either a Drill-Fix style rotary adaptor with an integral coolant ring must be used, or an end mill holder and a separate radial coolant ring are assembled together.



COMPETITIVE INSERT INTERCHANGEABILITY

The trigon inserts used in the Drill-Fix, HTS and HTS-C product lines can be utilized in some of our competitors tooling. For further details on this subject, see the cross-reference chart in Hertel Marketing Update numbers 88-1 (8/15/88) and 89-06 (7/24/89).

APPLICATION SCOPE

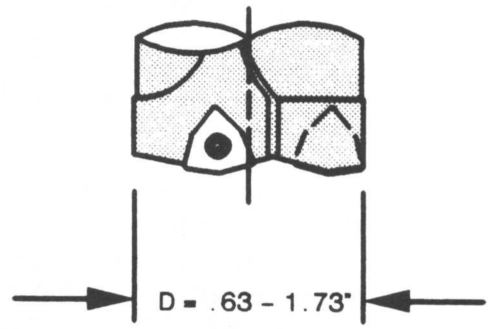
The Drill-Fix drill is designed for modern rigid machine tools (NC lathes and Machining Centers) and not radial drills, post drills or column type drilling machines. The carbide grades and positive cutting geometry of Drill-Fix is best suited for the machining of carbon steel, die steel, stainless steel, cast iron, aluminum, titanium and nickel based alloys. The Drill-Fix system will attain hole depths equal to 2.5 drill diameters.

PRODUCT SCOPE

Drill-Fix is available in 67 different standard drill sizes and five shank diameters. Let's examine the breakdown of this extensive program:

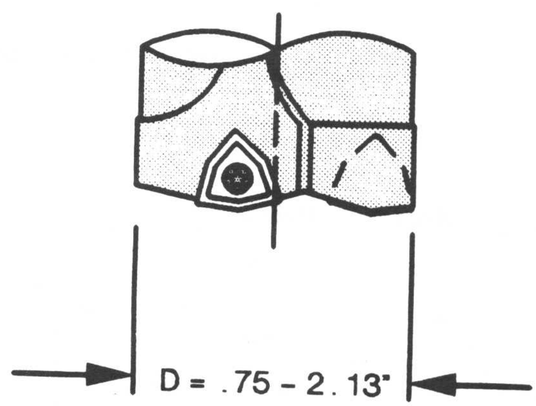
DRILL-FIX DIAMETER (.63"-1.73")

The Drill-Fix tools available in this diameter range can either be ordered with a 1.250" or 32mm (1.26") diameter shank. There are 32 standard Drill-Fix sizes available in this range with either one of the shanks mentioned. Drill-Fix drills are spaced 1mm or approximately .04" apart, providing a wide variety of size options for the end user.



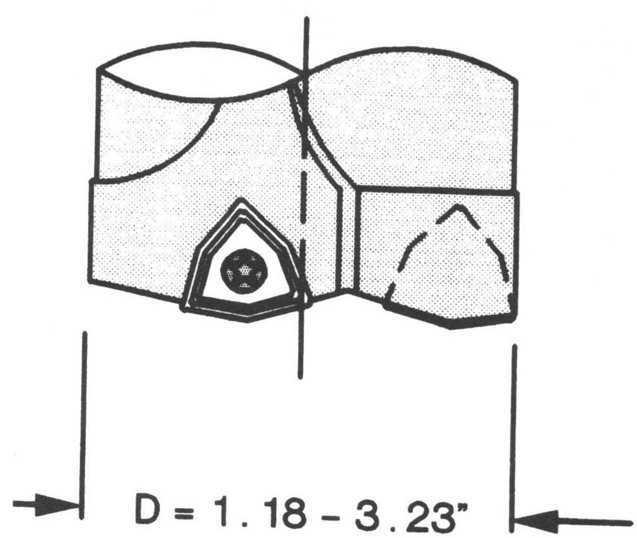
DRILL-FIX DIAMETER (.75"-2.13")

The Drill-Fix tools available in this diameter range can either be ordered with a 1.500" or 40mm (1.575") diameter shank. There are 36 standard Drill-Fix sizes available in this range with either one of the shanks mentioned.



DRILL-FIX DIAMETER (1.18"-3.23")

The Drill-Fix tools available in this diameter range can be ordered with a 50mm (1.969") diameter shank.



OPERATING PARAMETERS

The proper selection of feeds and speeds for the Drill-Fix is more complicated than the SE drill, since it has a range in recommended feeds and speeds for each workpiece material and drill size. The catalog operating parameters are provided in chart form (shown below). This speed and feed chart should be used to set the initial conditions for a drill test or demonstration. To establish the cutting speed, select a speed at the midpoint of the range shown. For example, if a customer wanted to drill a 1025 steel part with a 1.26" diameter drill, the initial cutting speed should be 650 SFPM.

The chart also shows a range of feed rates which can be selected to establish the starting parameters for Drill-Fix. The feed rate (inches/revolution or IPR) is provided as a range in the chart below, based on the drill diameter. The feed should also be the midpoint of the range specified as an initial starting point. In the previous example, a .005" IPR feed would be appropriate for a 1025 steel part and a 1.26" diameter drill. The strategy of selecting the midpoint in both speed and feed for an indexable drill, or any tool, is to prevent initial tool breakage due to a lack of system rigidity.

The following chart is shown in the Drill-Fix drill catalog:

Drill-Fix speeds & feeds

material group	tensile strength (psi)	BHN	material examples	recommended carbide grades	cutting speed	feed in inch per revolution for drill diamater				
						.63-.79	.83-1.18	1.22-1.73	1.77-2.13	2.17-3.23
unalloyed steels (C<.20%)	72,000	150	1015	CM4,CD4,CM3	500-1000	.002-.005	.003-.006	.003-.007	.004-.008	.004-.010
unalloyed steels (C.20%-.30%)	86,000	170	1025	CM4,CD4,CM3	500-800	.002-.005	.003-.006	.003-.007	.004-.008	.004-.010
unalloyed steels (C.30%-.40%)	100,000	200	1030	CM4,CD4,CM3	500-800	.002-.005	.003-.006	.003-.007	.004-.008	.004-.010
alloyed steels (C.30%-.40%)	94,000	185	4140	CM4,CD4,CM3	350-600	.002-.005	.003-.006	.003-.007	.004-.008	.004-.010
unalloyed steels (C.40%-.50%)	115,000	230	1045	CM4,CD4,CM3	380-650	.002-.005	.003-.006	.003-.007	.004-.008	.004-.010
alloyed steels (C.40%-.50%)	100,000	200	52100	CM4,CD4,CM3	300-600	.002-.005	.003-.006	.003-.007	.004-.008	.004-.010
alloyed steels	115,000	230	6150	CM4,CD4,CM3	350-500	.002-.005	.003-.006	.004-.008	.004-.010	.004-.012
alloyed steels	130,000	260	8620	CM4,CD4,CM3	350-500	.002-.005	.003-.006	.004-.008	.004-.010	.004-.012
alloyed steels	<144,000	290	H13	CM4,CD4,CM3	250-400	.002-.004	.002-.005	.003-.006	.004-.008	.004-.010
alloyed steels	>144,000	290	D3, D7	CM4,CD4,CM3	250-400	.002-.004	.002-.005	.003-.006	.004-.008	.004-.010
stainless, heat resistant steels	72,000	150	304, 410	CD4,KM1,KMF	300-600	.002-.004	.002-.005	.003-.006	.004-.008	.004-.010
stainless, heat resistant steels	86,000	170	420, 316	CD4,KM1,KMF	300-600	.002-.004	.002-.005	.003-.006	.004-.008	.004-.010
stainless, heat resistant steels	108,000	215	321	CD4,KM1,KMF	300-600	.002-.004	.002-.005	.003-.006	.004-.008	.004-.010
cast irons		150-200	G25	KM1	250-400	.003-.007	.004-.008	.004-.010	.004-.012	.004-.014
cast irons		200-220	G30	KM1	250-400	.003-.007	.004-.008	.004-.010	.004-.012	.004-.014
cast irons		220-250	G35	KM1	250-400	.003-.007	.004-.008	.004-.010	.004-.012	.004-.014
cast irons		250-320	G35	KM1	250-400	.003-.007	.004-.008	.004-.010	.004-.012	.004-.014
aluminum wrought alloys				KM1	600-900	.003-.007	.004-.008	.004-.010	.004-.012	.004-.014
aluminum alloys < 10% Si				KM1	900-1200	.003-.007	.004-.008	.004-.010	.004-.012	.004-.014
aluminum alloys > 10% Si				KM1	300-500	.003-.007	.004-.008	.004-.010	.004-.012	.004-.014

TROUBLESHOOTING GUIDE

The Drill-Fix indexable drill is a high performance drilling tool which often operates at feed rates requiring a great deal of horsepower. Therefore, it is essential that special attention is given to the rigidity of the components in the operation including the machine, workpiece and fixturing. Please review the following recommendations which are designed to identify and eliminate common problems encountered when using this tool.

CHIP EVACUATION

Drill-Fix requires a coolant supply at a pressure of 45 psi and a volume of 5-6.5 gallons/minute. Coolant is used to evacuate chips and dissipate heat; therefore, the greater the coolant pressure and volume, the cooler the part and the longer the tool life.

CHIP FORMATION

If long stringy flat chips are a problem, increase the cutting speed first, while leaving the feed/revolution constant. Chips should never turn dark blue in color. If this occurs, either reduce cutting speed, or increase the coolant supply. When chip formation is not satisfactory after raising the cutting speed, then begin increasing the feed/revolution until adequate chip formation takes place.

BREAKAGE OR CHIPPING OF THE CENTER INSERT

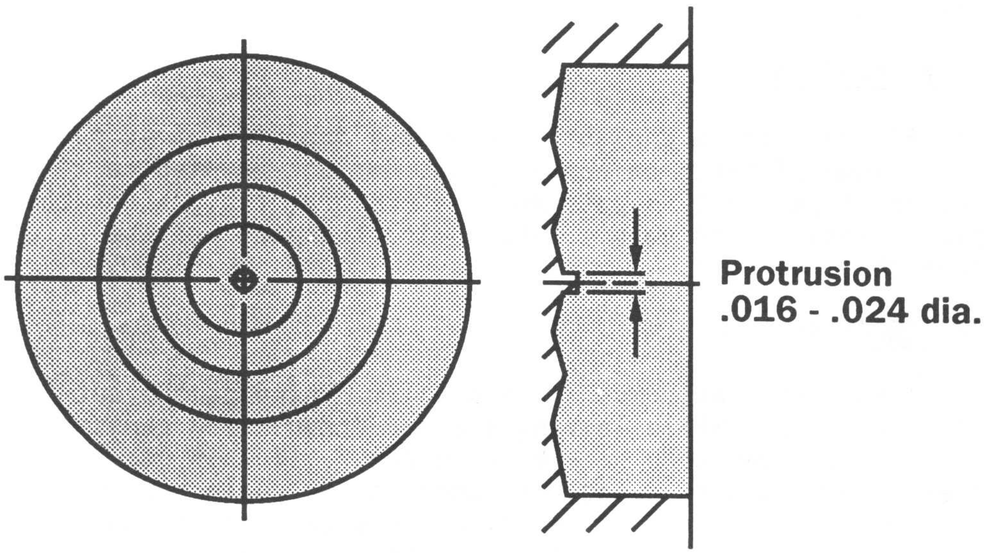
The cutting speed at the centerline of any drill is always zero; therefore, this area of the tool must be durable enough to survive the compressive force of the machine feed without breaking. For this reason, the in-board or center insert of an indexable drill should be tougher or uncoated, while the exterior insert often must combat the excessive cutting speed typical of this type of operation. Hence, it is often a coated carbide grade. Two carbide grades are often used in the same drill simultaneously.

FEED SELECTION

In most cases, Drill-Fix is available with two insert stations. The catalog feeds are provided in inches/revolution, since all indexable drills currently on the market have one effective cutting flute, regardless of the number of insert stations/tool.

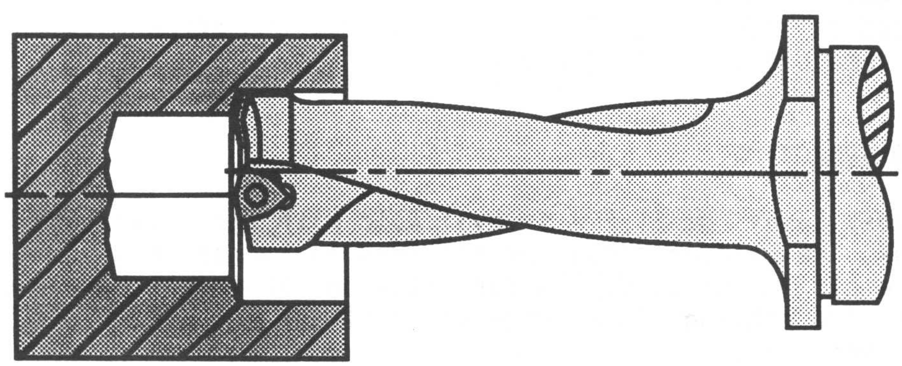
DRILL TO CENTERLINE RELATIONSHIP

The alignment of Drill-Fix on a lathe is critical to the performance of the tool. The in-board insert of Drill-Fix should be .008"-.012" below the machine centerline to insure the appropriate clearance is maintained on the tool. This can be established by drilling a short distance into the part and checking the hole bottom for a protrusion of .016"-.024". See picture:



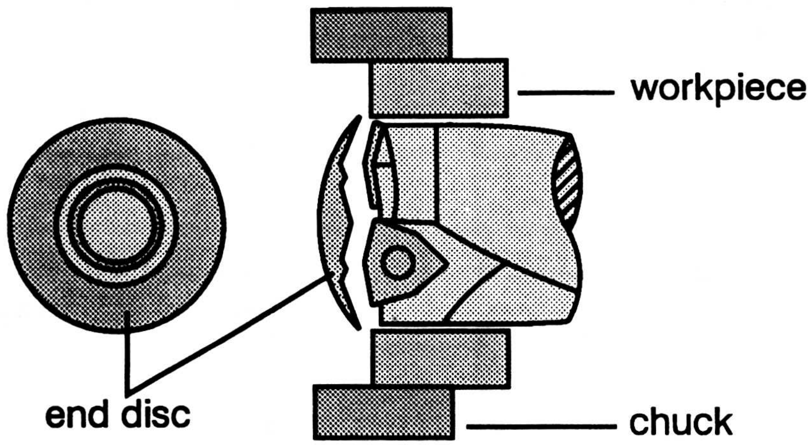
BORING WITH DRILL-FIX

When Drill-Fix is used as a boring tool, it must be offset in the direction of the outboard insert. This is also true when Drill-Fix is used to offset drill from a solid.



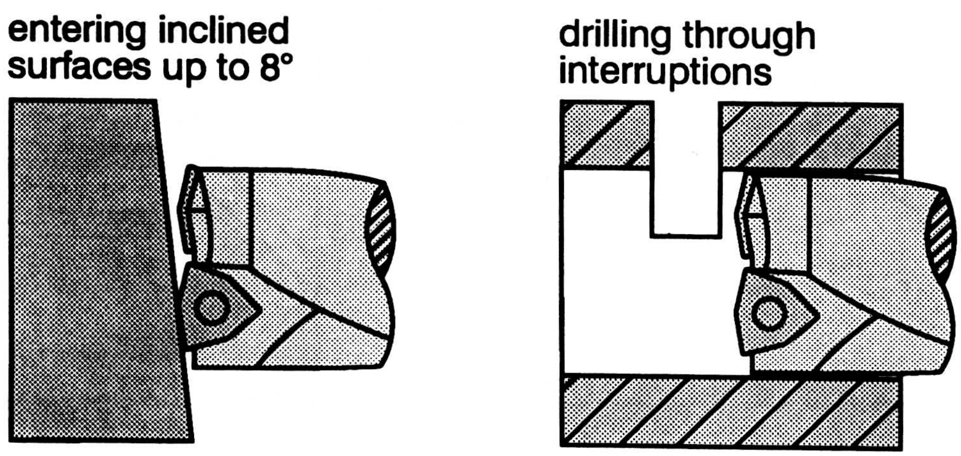
DISK FORMATION

When Drill-Fix is applied on a lathe or non-rotating application, a disk is created and discharged as the tool exits the workpiece. This disk is ejected at high velocity and poses a significant hazard if the machine is not properly enclosed. Appropriate precautions should be taken under these circumstances to limit the danger to the machine operator.



INCLINED SURFACES AND INTERRUPTIONS

Drill-Fix is capable of drilling surfaces with up to an eight degree angle of inclination. In addition, Drill-Fix is capable of drilling through interruptions as long as the inboard insert remains engaged in the workpiece.

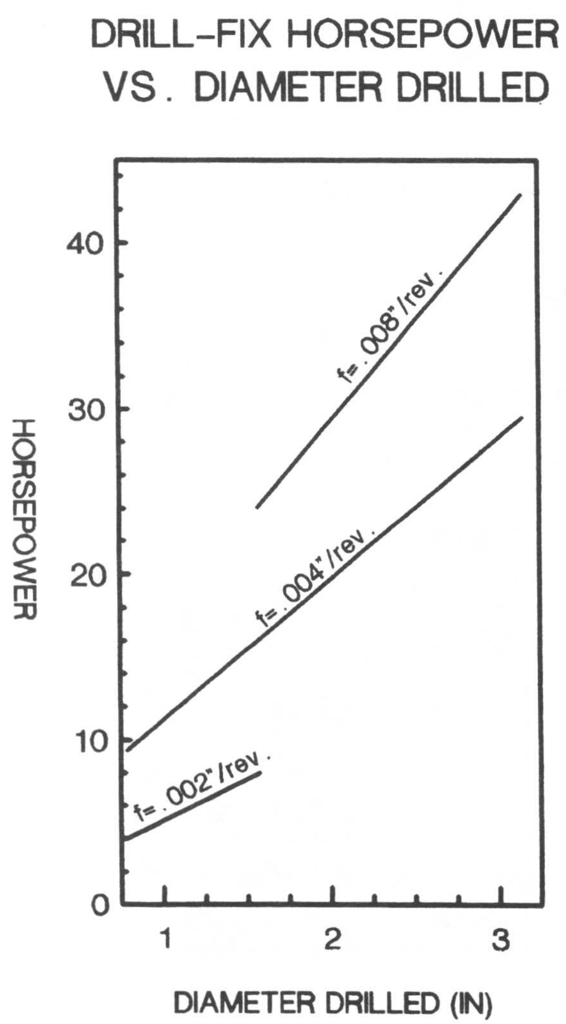


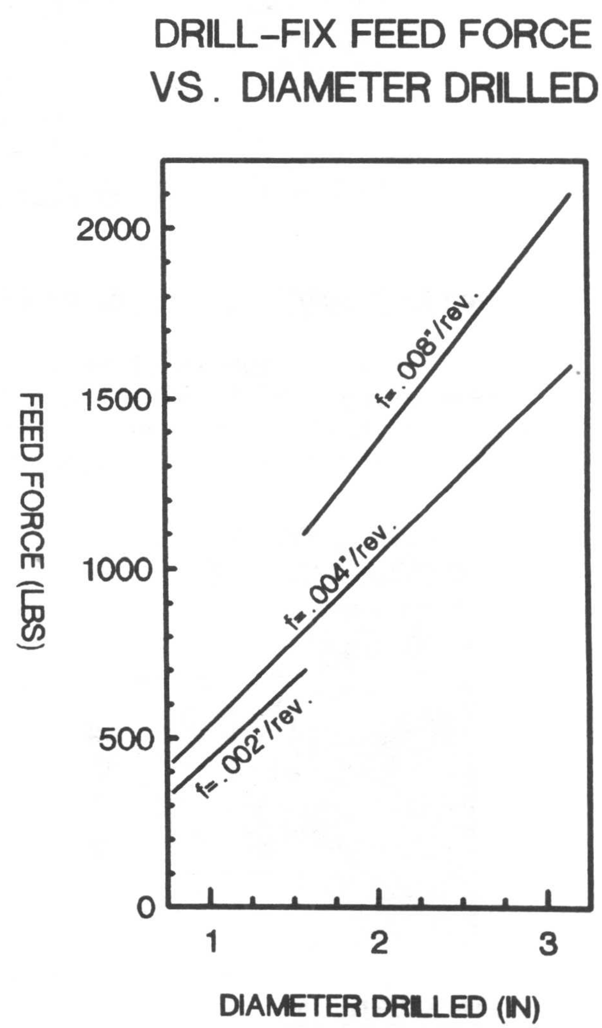
STACK DRILLING

Drill-Fix should not be used to drill through plates of stacked material, one on top of another, because as the tool exits each piece a disk is created. Since the disk is not ejected, but trapped between the plates, it will cause catastrophic failure of the drill.

FEED FORCE AND HORSEPOWER

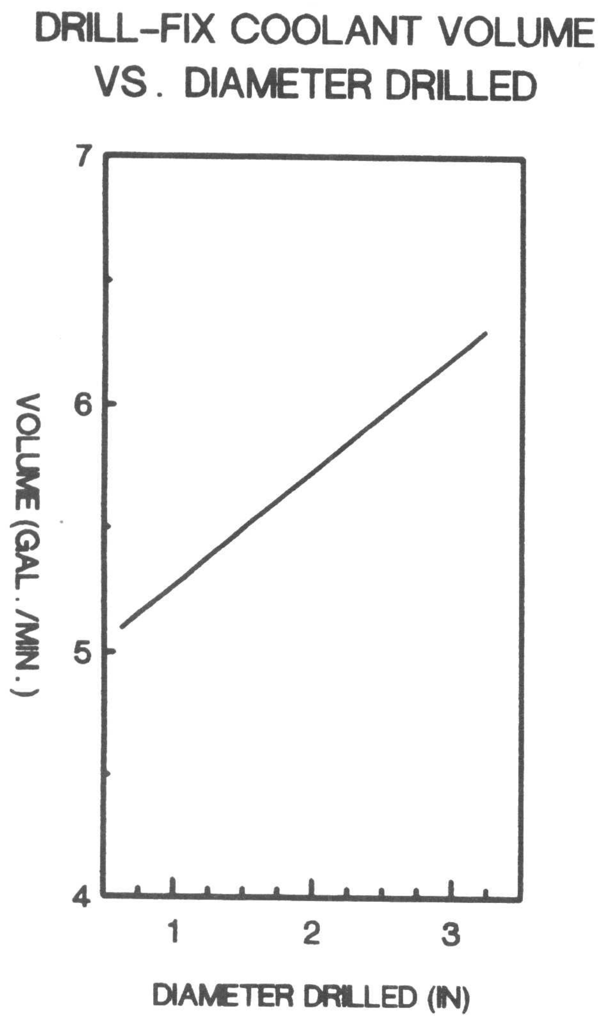
The graphs shown below illustrate the feed force produced and power consumed when drilling a 116,000 psi high carbon steel at 300 SFPM. Three different feed rates are provided on each graph to allow selection of the appropriate cutting conditions in advance of actually running a Drill-Fix on a low power machine tool. Therefore, if a question arises as to whether a prospective machine tool has adequate power or thrust capability to run a Drill-Fix, the charts can be used to make that determination.

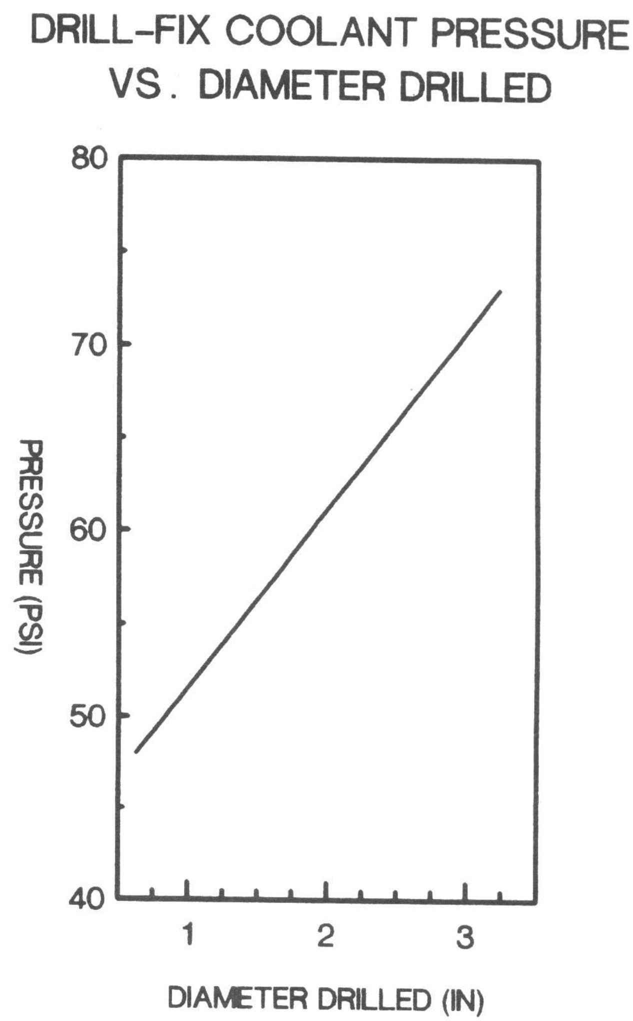




COOLANT REQUIREMENTS

The graphs shown below give specific information for the required coolant volume and pressure necessary to drill a hole 2.5 diameters in depth with Drill-Fix:





In those instances where a hole depth of one diameter is required, external flood coolant is normally sufficient.

FIELD TEST RESULTS

The following tests were completed using Drill-Fix indexable drills. These results have all led to purchase orders by each customer:

Drill Diameter	RPM	Cutting Speed SFPM	Feed IPR	Rate IPM	Hole Depth	Material	# of Holes
.98"	1200	308	.005	6	.81"	D2 die steel	NA
1.02"	4300	1150	.002	8.6	2.20"	1045 steel	300
1.46"	1280	490	.005	6.4	2.75"	6150 steel	70
1.14"	2210	660	.003	6.6	1.90"	304 S.S.	90
1.34"	600	210	.002	1.2	1.10"	Titanium alloy	20
2.95"	1811	1400	.001	1.8	6.69"	Electrolytic Copper	25

THE HTS & HTS-C MODULAR DEEP HOLE DRILLS

BACKGROUND

Indexable drills have been proven to be extremely unstable as they increase in drilling length beyond three diameters. At four diameters in flute length, the tool has 20% of the rigidity it had at two diameters. In specific applications, carbide wear pads and high helix drills have been designed to eliminate excessive deflection as the tool length is increased. However, these individual solutions are often expensive specials which require lead time for manufacturing. Hertel has developed two products which address the need for standard indexable deep hole drilling systems. The Hertel drilling product designated HTS (HERTEL-Tiefbohr-System = HERTEL deep-hole drilling system) can produce holes eight diameters in hole depth, while HTS-C (HERTEL-Tiefbohr-System-Compact = HERTEL deep-hole drilling system-Compact) is capable of hole depths either three or five diameters deep.

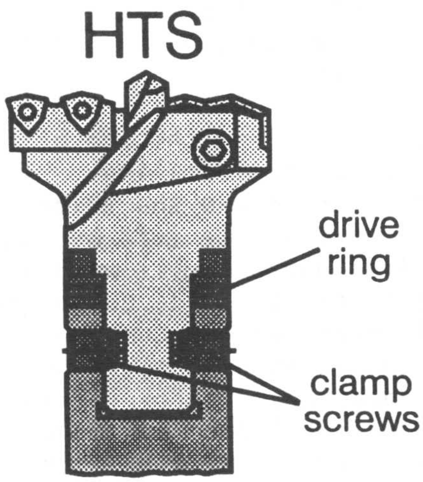
These two products obtain their stability from the use of a HSS center-drill, which acts as a guide when drilling. HTS utilizes an inner and outer cartridge to secure the trigon inserts used to drill the remainder of the hole left by the pilot drill. However, HTS-C is a fixed pocket drill with a trigon in-board station and a square insert in the out-board position. Both drill styles have interchangeable components. HTS has separate modular heads, shanks, extensions, reducers and adaptors, while HTS-C has two shank options for (three or five times the head diameter in hole depth) each drill head. The Hertel deep hole product lines offer reliability in high performance drilling from .787" (20mm)-6.69" (162mm) in diameter.

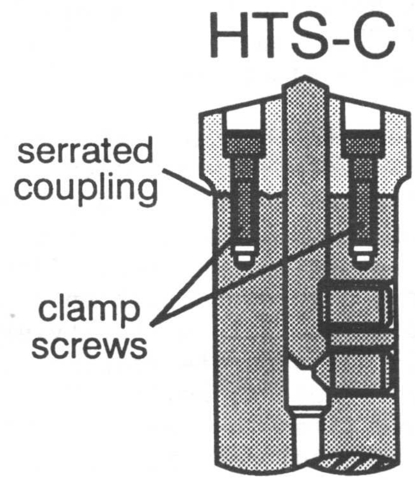
PRODUCT FEATURES

HTS and HTS-C share many common product features which have been incorporated in these tools to enhance their flexibility and productivity. These tools are capable of drilling holes from a solid workpiece of three to eight diameters deep depending on the tool style and diameter selected. Therefore, HTS and HTS-C are considered deep hole indexable drilling systems. Let's examine the unique features of these product lines:

UNIQUE DRIVE SYSTEMS

HTS and HTS-C have drive systems between individual components which are both unique and very stable when subjected to normal cutting loads. For example, HTS heads, extensions reducers and basic shanks are linked together using two opposing offset conical set screws which secure the assembly, and a replaceable drive ring that functions as protection for the individual components in the event of a major failure. The HTS-C drilling system has a serrated joint between the drill head and shank. The head, in this case, is secured on the shank using socket head cap screws. These two drive systems transmit the necessary torque to allow both modular drilling systems to operate at peak performance.

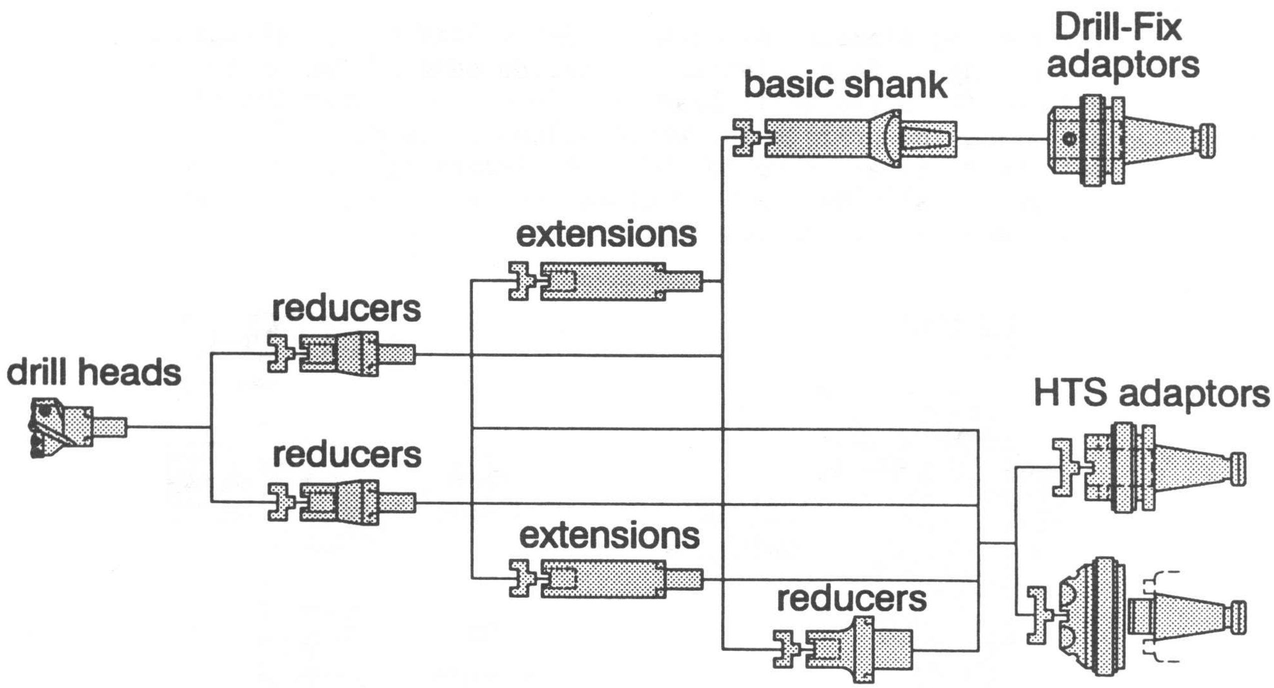




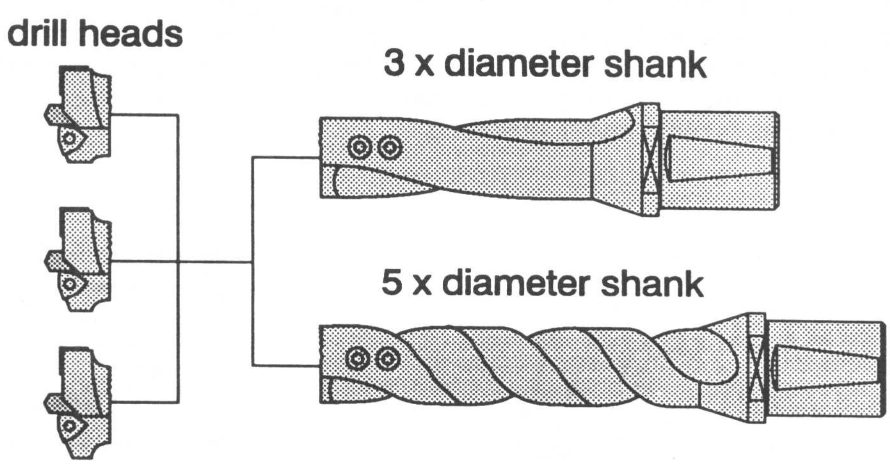
FLEXIBILITY

The HTS and HTS-C drill systems were designed with modular components to maximize the flexibility of these individual product lines. The HTS drill head can be mounted directly in an adaptor, extension, reducer or basic shank. In addition, the same head could be part of a combined assembly of all the components mentioned. HTS-C has the option of two different drilling depths (by making a shank change, 3xD or 5xD) for each drill head diameter, or up to six different drill head diameters per drill shank. (See next page)

HTS is designed for maximum flexibility

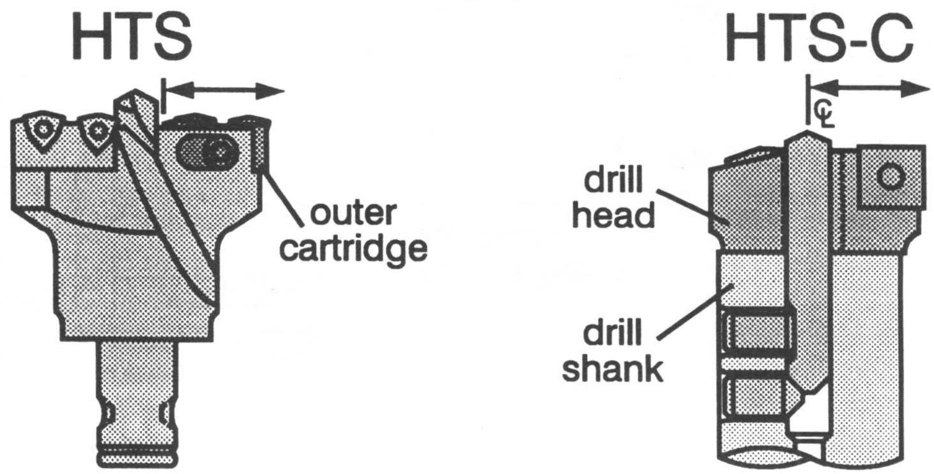


HTS-C offers two different shank options



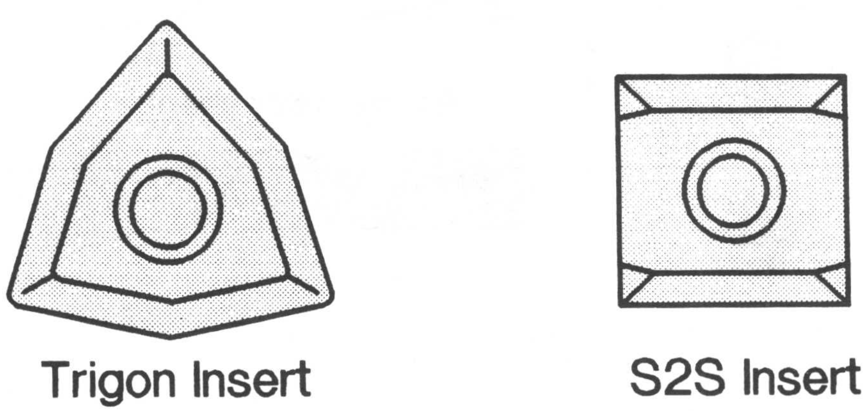
DIAMETER ADJUSTMENT

The cutting diameter of each HTS drill head can be changed up to .2", by surface grinding the inside edge of the outer cartridge. The HTS-C drill head has clearance between the pilot drill and the drill head, which allows adjustment of the cutting diameter of up to .024", by loosening the mounting screws and sliding the head along the serrations toward the out-board insert station.



OPTIONAL CHIP BREAKERS & INSERTS

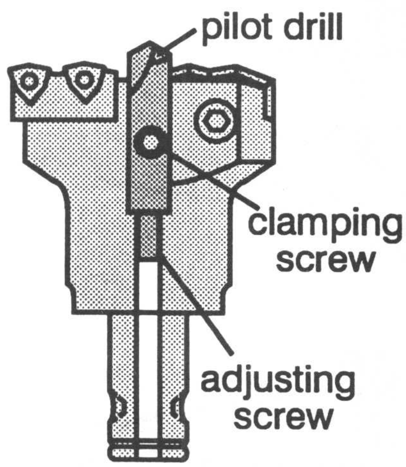
The HTS system utilizes trigon inserts which can either be ordered with standard chip breakers or more positive rake "24" style chip breakers for high nickel, low carbon steel and titanium workpiece materials. Trigon inserts can also be ordered, in either standard or heavy duty (thicker) versions, for use in the HTS drill heads after changing insert cartridges. The HTS-C drill utilizes the trigon insert on the in-board position and an S2S style insert in the out-board station. The S2S insert has three optional chip breakers for aluminum, steel and cast iron.



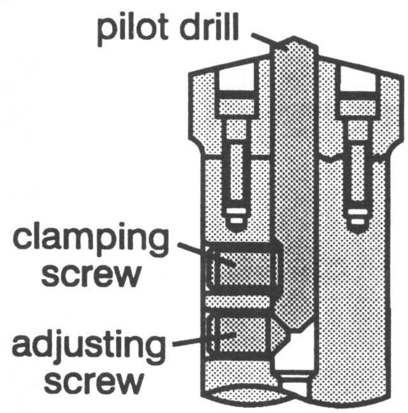
ADJUSTABLE PILOT DRILLS

The pilot drill used on both the HTS and HTS-C modular drilling systems can be adjusted to accommodate changes in either machining conditions or length alterations due to resharpening. The HTS pilot drill is adjusted from the rear of the drill head via an axial adjusting screw. Once the appropriate length is established, the drill is secured in position using the set screw located on the side of the drill head. The HTS-C pilot drill can be adjusted with the head mounted on the drill shank, using the side adjusting screw. The HTS-C pilot drill is also secured using a radial clamping screw. See the HTS and HTS-C catalogs regarding the appropriate pilot drill set length.

HTS



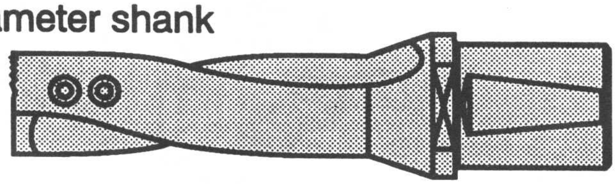
HTS-C



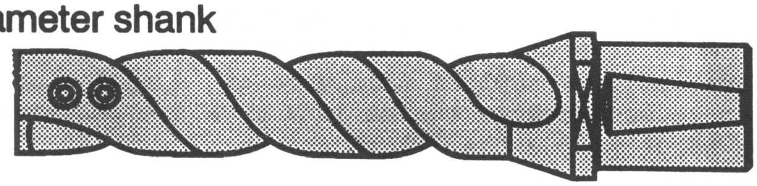
DRILL SHANK OPTION

The HTS-C modular drilling system has two shank styles available. The first has the capability to drill three drill diameters deep using a slow helix (25 degree) type flute. The second will drill holes up to five drill diameters deep, with a fast helix (45 degree) for added rigidity and enhanced chip evacuation.

3 x diameter shank



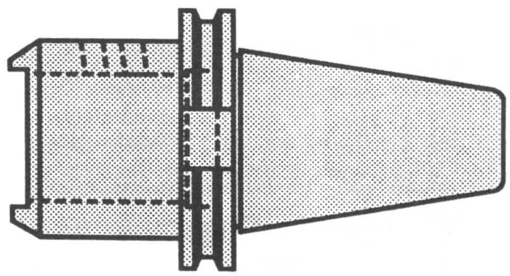
5 x diameter shank



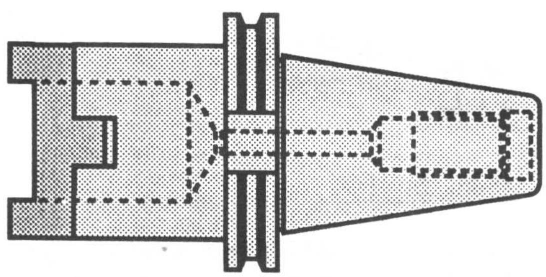
EXTENSIVE ADAPTOR PROGRAM

The HTS system has a significant number of adaptor styles available to meet a variety of machine needs. HTS style adaptors are available in CAT 50 & NMTBA 50 machine tapers, with or without coolant rings. The HTS style adaptor can be coupled directly with a drill head, extention or reducer. The Drill-Fix style adaptor with drive tangs is available in CAT 40, CAT 50, NMTBA 50, Morse 4 and Morse 5 tapers, with coolant rings in most cases. The Drill-Fix style adaptor is designed to couple directly with HTS basic shanks. The HTS program includes an adaptor (flange mounted) which can be bolted directly to the spindle face of a machine, using a four-inch bolt circle. This adaptor is designed to be coupled directly to a drill head, extension or reducer.

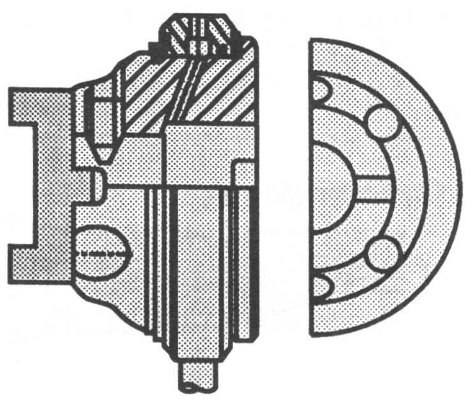
Drill-Fix style
tapered adaptor



HTS style
tapered adaptor

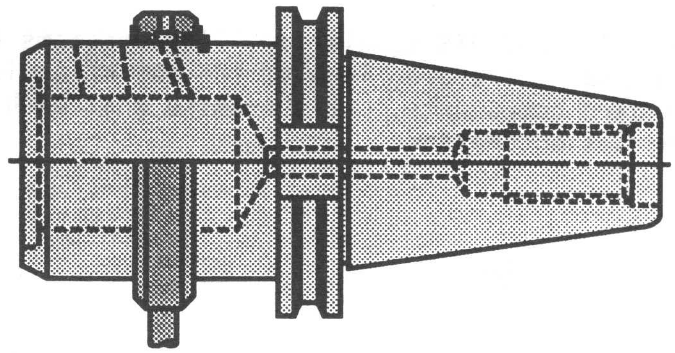


HTS style
flanged adaptor

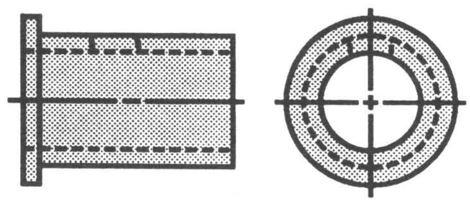


HTS-C utilizes Cat 40 & 50, BT 40 & 50 taper adaptors as well as NMTBA 50 adaptors. In addition morse taper adaptors are also available to support this product. A series of bushins is used to accomodatee the variety of HTS-C shank sizes in a single adaptor. See the illustrations on page 52:

HTS-C adaptor



HTS-C bushing



APPLICATION SCOPE

The HTS modular indexable deep hole drilling system is often a direct replacement for existing HSS or carbide spade drills on lathes, VTLs or boring mills. This tool is capable of attaining drilled hole depths equal to eight times its drill head cutting diameter. The standard product line is designed to drill holes from 1.77"- 6.69" in diameter. This system has the flexibility to utilize thick trigon inserts for heavy duty applications by simply changing cartridges. HTS requires rigid machine tools, not radial drills, drill presses or small vertical milling machines.

HTS-C is designed to displace HSS twist drills on high performance NC lathes and machining centers. This product is available in a .78"-1.77" diameter range for steel, stainless steel, cast iron, high temperature alloy and aluminum workpieces. The standard HTS-C drill has the capability to drill holes three or five drill head diameters in depth. This product should not be applied on non-rigid machine tools, such as drill presses, radial drills or small vertical mills.

The pilot drill life, of both HTS and HTS-C, can be greatly enhanced by any improvements in coolant pressure and volume. The Hertel coolant specifications are a minimum requirement to evacuate chips. Therefore, any increases beyond these levels will act to reduce friction and heat in the cutting zone and hence, increase the life of the tool.

PRODUCT SCOPE

The HTS-C and HTS modular drilling programs are quite extensive and, therefore, involve the use of a wide variety of standard components. Let's examine what each of these programs encompasses:

ITEM	HTS-C	HTS
Drilling Diameter Range	.787"-1.77"	1.77"-6.69"
Drilling Depth	3 or 5 diameters	up to 8 diameters
# of Standard Heads	26	20
# of Standard Shanks	7 Short (3 x D) 7 Long (5 x D)	22 basic 9 extensions 9 reducers
# of Pilot Drills	4	6 without holes 4 with holes
# of Adaptors	1 CAT 40 1 CAT 50 2 NMTBA 2 Morse 2 BT	1 CAT 40 11 CAT 50 4 NMTBA 4 Morse 1 Flanged
# of Insert Sizes	3 Trigon 4 S2S	5 Thin Trigon 4 Thick Trigon

OPERATING PARAMETERS

The proper selection of feeds and speeds for the HTS & HTS-C systems is more complicated than the SE drill, since they have a range in recommended feeds and speeds for each workpiece material and drill size. The catalog operating parameters are provided in chart form (shown below and on the next page). These speed and feed charts should be used to set the initial conditions for a drill test or demonstration. To establish the cutting speed, select a speed at the midpoint of the range shown. For example, if a customer wanted to drill a 1025 steel part with a 6.69" diameter HTS drill, the initial cutting speed should be 330 SFPM.

The charts also show a range of feed rates which can be selected to establish the starting parameters for both systems. The feed rate (inches/revolution or IPR) is provided as a range in the charts, based on the drill diameter. The feed should also be the midpoint of the range specified as an initial starting point. In the previous example, a .008" IPR feed would be appropriate for a 1025 steel part and a 6.69" diameter drill. The strategy of selecting the midpoint in both speed and feed for a modular deep hole indexable drill, or any tool, is to prevent initial tool breakage due to a lack of system rigidity. In the case of a HTS or HTS-C, this approach allows you to evaluate the amount of wear on the pilot drill (normally the weakest link in the system) at a moderate cutting speed. Note: just as there were cutting speed adjustments as the hole depth changed with SE drills, the same practice is performed for HTS (see chart below):

HTS speeds & feeds

material group		tensile strength (psi)	BHN	material examples	recommended carbide grades	cutting speed	feed in inch per revolution for drill diamater range				
							1.75-2.25	2.25-3.00	3.00-4.00	4.00-4.80	4.80-6.69
unalloyed steels	(C<.20%)	72,000	150	1015	CM4,CD4,CM3	260-400	.003-.005	.004-.006	.005-.007	.006-.010	.006-.012
unalloyed steels	(C.20%-.30%)	86,000	170	1025	CM4,CD4,CM3	260-400	.003-.005	.004-.006	.005-.007	.006-.010	.006-.012
unalloyed steels	(C.30%-.40%)	100,000	200	1030	CM4,CD4,CM3	260-400	.003-.005	.004-.006	.005-.006	.006-.008	.006-.010
alloyed steels	(C.30%-.40%)	94,000	185	4140	CM4,CD4,CM3	260-400	.003-.005	.004-.006	.005-.006	.006-.008	.006-.010
unalloyed steels	(C.40%-.50%)	115,000	230	1045	CM4,CD4,CM3	230-330	.002-.004	.003-.005	.004-.006	.005-.006	.006-.008
alloyed steels	(C.40%-.50%)	100,000	200	52100	CM4,CD4,CM3	230-330	.002-.004	.003-.005	.004-.006	.005-.006	.006-.008
alloyed steels		115,000	230	6150	CM4,CD4,CM3	230-330	.002-.004	.003-.005	.004-.006	.005-.006	.006-.008
alloyed steels		130,000	260	8620	CM4,CD4,CM3	200-300	.002-.003	.003-.004	.004-.006	.005-.006	.005-.006
alloyed steels		<144,000	290	H13	CM4,CD4,CM3	200-260	.002-.003	.002-.004	.003-.005	.004-.006	.005-.006
alloyed steels		>144,000	290	D3, D7	CM4,CD4,CM3	160-260	.002-.003	.002-.003	.002-.004	.003-.005	.003-.005
stainless, heat resistant steels		72,000	150	304, 410	CD4,KM1,KMF	200-330	.002-.004	.003-.005	.003-.005	.004-.006	.004-.006
stainless, heat resistant steels		86,000	170	420, 316	CD4,KM1,KMF	200-300	.002-.003	.002-.004	.002-.004	.003-.005	.004-.006
stainless, heat resistant steels		108,000	215	321	CD4,KM1,KMF	200-260	.002-.003	.002-.003	.002-.003	.003-.004	.004-.005
cast irons			150-200	G25	KM1	260-400	.004-.006	.005-.008	.006-.010	.006-.010	.006-.012
cast irons			200-220	G30	KM1	260-330	.004-.006	.005-.008	.006-.010	.006-.010	.006-.012
cast irons			220-250	G35	KM1	230-300	.003-.005	.004-.006	.005-.008	.006-.008	.006-.010
cast irons			250-320	G35	KM1	200-260	.003-.005	.004-.006	.005-.008	.005-.008	.006-.010
aluminum wrought alloys					KM1	600-800	.002-.004	.002-.004	.003-.005	.004-.006	.005-.008
aluminum alloys < 10% Si					KM1	600-800	.002-.004	.003-.005	.004-.006	.005-.008	.005-.008
aluminum alloys > 10% Si					KM1	200-300	.002-.004	.003-.005	.004-.006	.005-.008	.005-.008

The standard values indicated for cutting speeds should be adjusted according to drilling depth and diameter. Reference your drilling diameter and depth/diameter ratio in the table at right to find the appropriate correction factor. Multiply this factor by the recommended cutting speed from the table above.

adjustment factors per drill diameter range and drilling depth ratio

1.75-3.00		3.00-4.80		4.80-6.69	
ratio	factor	ratio	factor	ratio	factor
4 x d_1	1.2	3 x d_1	1.1	2 x d_1	1.0
6 x d_1	1.0	4.5 x d_1	1.0	3.5 x d_1	.9
8 x d_1	.8	6 x d_1	.8	5 x d_1	.8

The following chart is shown in the HTS-C drill catalog:

HTS-C speeds & feeds

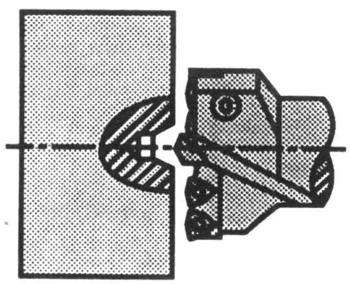
material group		tensile strength (psi)	BHN	material examples	recommended carbide grades	cutting speed 3 x D	cutting speed 5 x D	feed in inch per revolution for drill diamater range .78-.83	.83-1.10	1.10-1.35	1.35-1.55	1.55-1.78
unalloyed steels	(C<.20%)	72,000	150	1015	CM4,CD4,CM3	656	459	.002	.0025	.003	.004	.004
unalloyed steels	(C.20%-.30%)	86,000	170	1025	CM4,CD4,CM3	590	426	.0025	.003	.004	.005	.005
unalloyed steels	(C.30%-.40%)	100,000	200	1030	CM4,CD4,CM3	557	393	.003	.0035	.004	.005	.005
alloyed steels	(C.30%-.40%)	94,000	185	4140	CM4,CD4,CM3	557	393	.003	.0035	.004	.005	.005
unalloyed steels	(C.40%-.50%)	115,000	230	1045	CM4,CD4,CM3	492	328	.0025	.003	.0035	.0045	.0055
alloyed steels	(C.40%-.50%)	100,000	200	52100	CM4,CD4,CM3	292	328	.0025	.003	.0035	.0045	.0055
alloyed steels		115,000	230	6150	CM4,CD4,CM3	459	328	.002	.0025	.003	.004	.0045
alloyed steels		130,000	260	8620	CM4,CD4,CM3	393	262	.002	.002	.0025	.0035	.004
alloyed steels		<144,000	290	H13	CM4,CD4,CM3	328	229	.002	.002	.0025	.003	.0035
alloyed steels		>144,000	290	D3, D7	CM4,CD4,CM3	262	197	.002	.002	.0025	.003	.0035
stainless, heat resistant steels		72,000	150	304, 410	CD4,KM1,KMF	295	197	.002	.002	.0025	.003	.0035
stainless, heat resistant steels		86,000	170	420, 316	CD4,KM1,KMF	229	164	.002	.002	.002	.003	.003
stainless, heat resistant steels		108,000	215	321	CD4,KM1,KMF	197	137	.0015	.0015	.002	.002	.002
cast irons			150-200	G25	KM1	492	328	.004	.005	.0055	.0065	.007
cast irons			200-220	G30	KM1	459	328	.004	.005	.0055	.0065	.007
cast irons			220-250	G35	KM1	393	262	.003	.004	.0045	.005	.006
cast irons			250-320	G35	KM1	328	229	.003	.0035	.0045	.005	.0055
aluminum wrought alloys					KM1							
aluminum alloys < 10% Si					KM1							
aluminum alloys > 10% Si					KM1							

TROUBLESHOOTING GUIDE

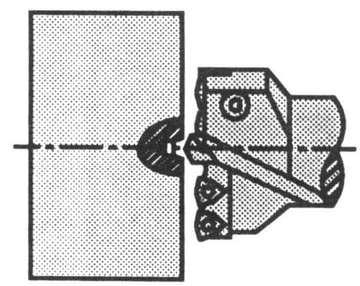
As the ratio of hole depth to tool diameter increases, every phase of a drilling operation becomes more critical to the successful application of the tool. Rigidity, coolant pressure, chip control, cutting speed, feed rate, workpiece hardness, workpiece configuration and the machine power are the application criteria which individually have the potential to determine whether a drilling operation succeeds or fails. The modular deep hole indexable drilling systems, known as HTS and HTS-C, therefore have some stringent application guidelines which must be followed. Let's examine these rules below and on the next few pages:

INCORRECT **CORRECT**

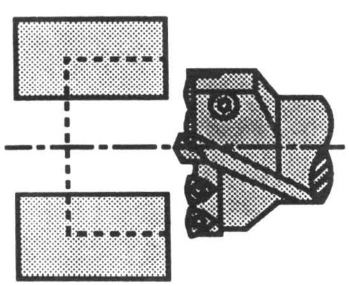
CENTER DRILLED HOLES



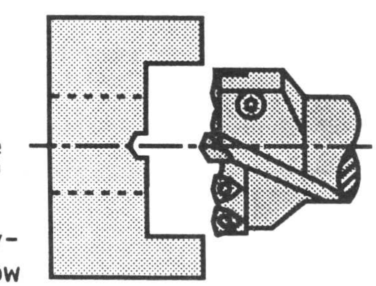
The pilot drill diameter of an HTS or HTS-C drill must be larger than the pre-centered hole diameter of a workpiece. If this requirement is ignored, the pilot drill will not properly guide the drill head through the hole.



STEPPED BORES

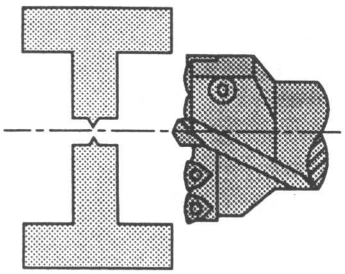


When drilling a series of stepped bores with HTS or HTS-C, always drill the largest bore first, so the succeeding drills will have material left at the hole center for the pilot drill. The pilot drill on the second drill should be the same diameter and .03" further ahead of the outboard insert than the previous drill. This will allow the pilot drill to actually cut the part before the inserts in the drill head.

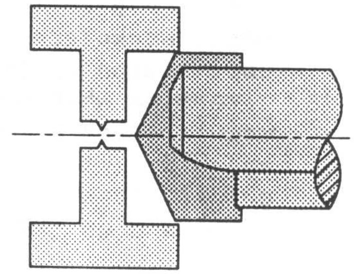


INCORRECT **CORRECT**

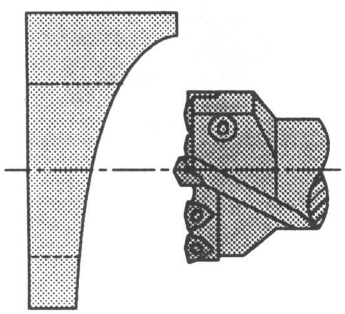
DRILLING FROM 2 SIDES



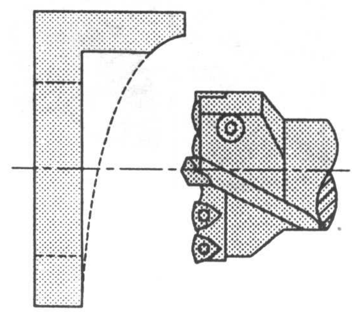
When drilling from two sides, the second drill should not be allowed to break through the part. The HTS or HTS-C pilot drill will lose contact upon break-through. When this situation arises, a spade drill is often used to remove the material at the hole center.



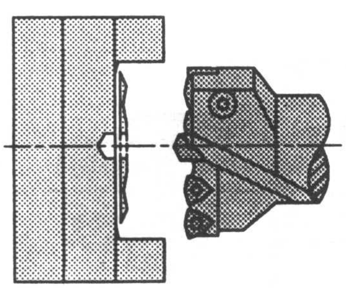
INCLINED SURFACES



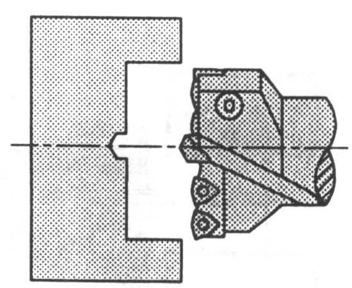
HTS-C drills can be used on surfaces with up to an eight degree entry angle, but only a four degree exit angle. However, since HTS is often a longer, and therefore less rigid assembly, it should be used only on flat entry and exit work surfaces.



STACK DRILLING

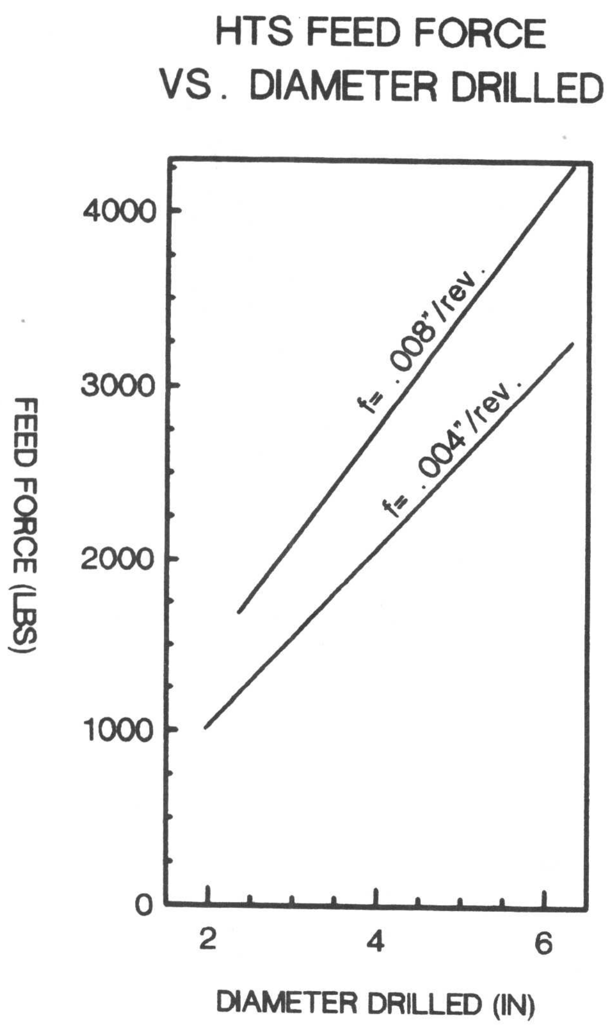


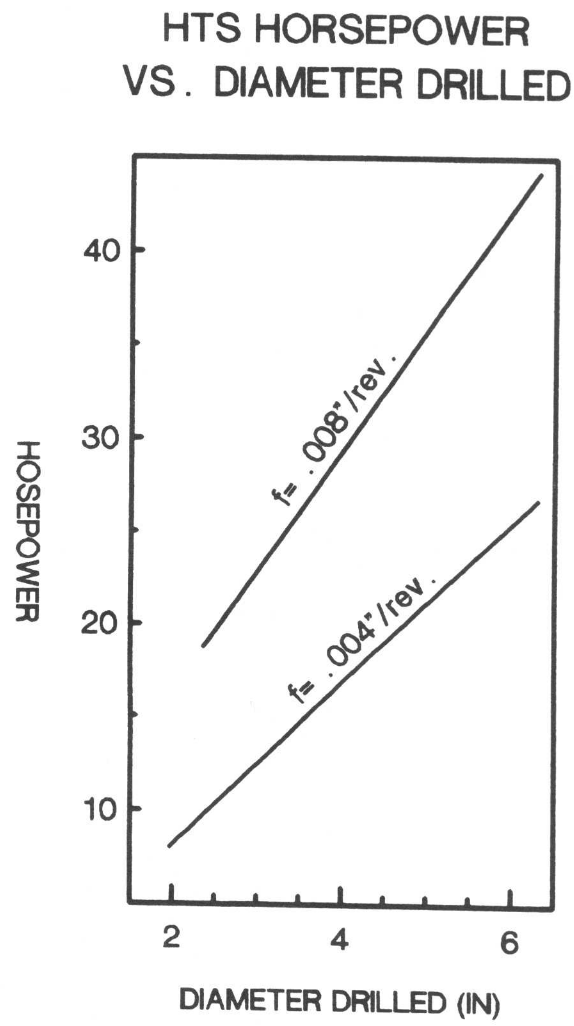
HTS and HTS-C should not be used to drill through plates of stacked material, one on top of another, because, as the tool exits each piece of plate, a disk is created. These tools should be used only to drill through holes on individual workpieces.



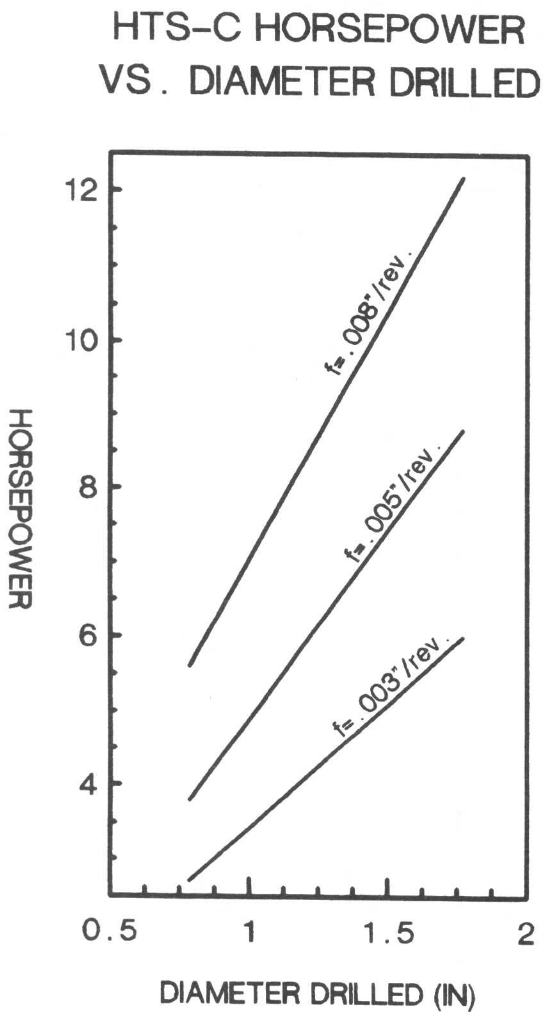
FEED FORCE AND HORSEPOWER

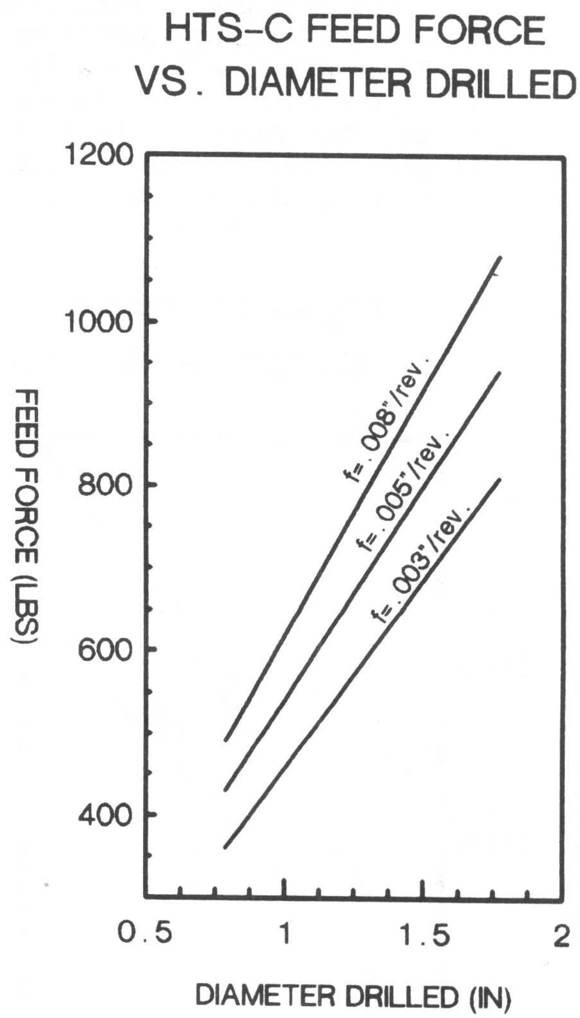
The graphs shown below illustrate the feed force produced and power consumed, when drilling a 87,000-130,000 psi carbon steel at 328 SFPM. Two (HTS) or three (HTS-C) different feed rates are provided on each graph to allow selection of the appropriate cutting conditions in advance of actually running an HTS or HTS-C drill on a low power machine tool. Therefore, if a question arises as to whether a prospective machine tool has adequate power or thrust capability to run HTS or HTS-C, the charts can be used to make that determination. The feed force produced and power required for HTS is as follows:





Horsepower consumption is dependant on the metal removal rate of an individual machining operation. Therefore, since an HTS drill has a larger diameter than an HTS-C style drill, you should expect a greater metal removal rate and minimum machine horsepower requirement for HTS tools when compared to the HTS-C modular deep hole drilling system. The feed force produced by HTS will also be greater than HTS-C as illustrated by the HTS-C feed force and horsepower graphs shown below:

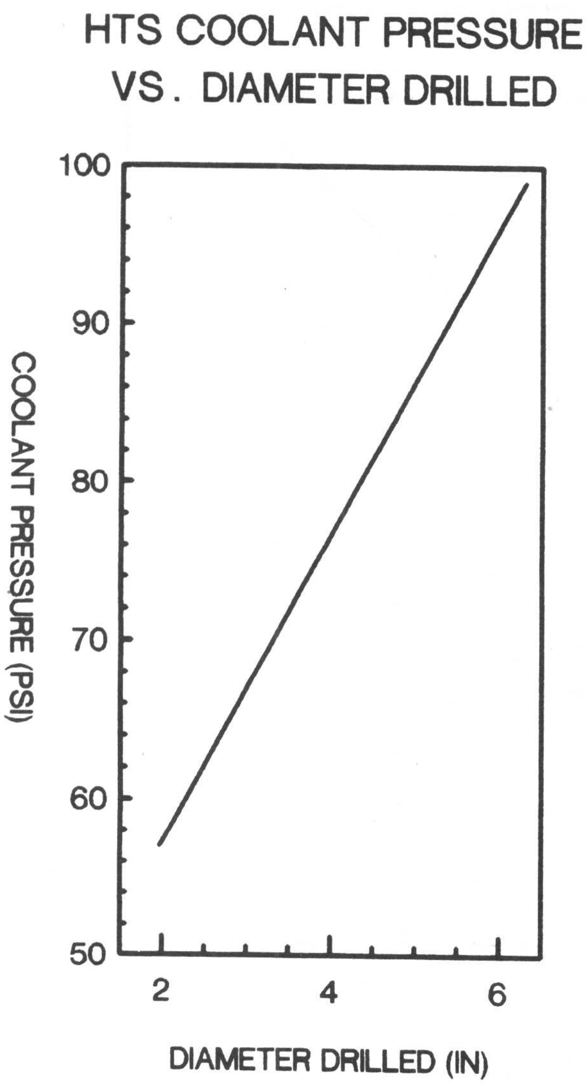


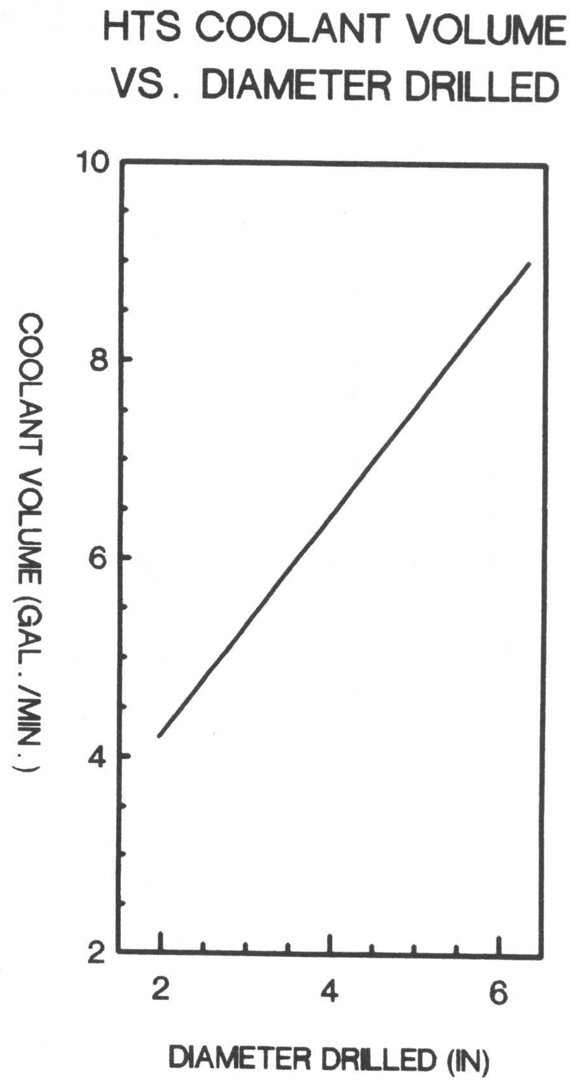


The HTS drilling system requires machines in the range of 8-40 horsepower while the HTS-C drilling system can be utilized on machines of 12 horsepower or less. The feed forces produced with HTS drills are from 1000-4300 lbs. depending on the drilled hole diameter and feed rate. HTS-C feed forces are less than 1300 lbs. Therefore, the machine requirements in terms of rigidity and power are significantly different when comparing the HTS and HTS-C modular deep hole drilling systems.

COOLANT REQUIREMENTS

The graphs shown below give specific information for the required coolant volume and pressure necessary to drill a hole with HTS is as follows:

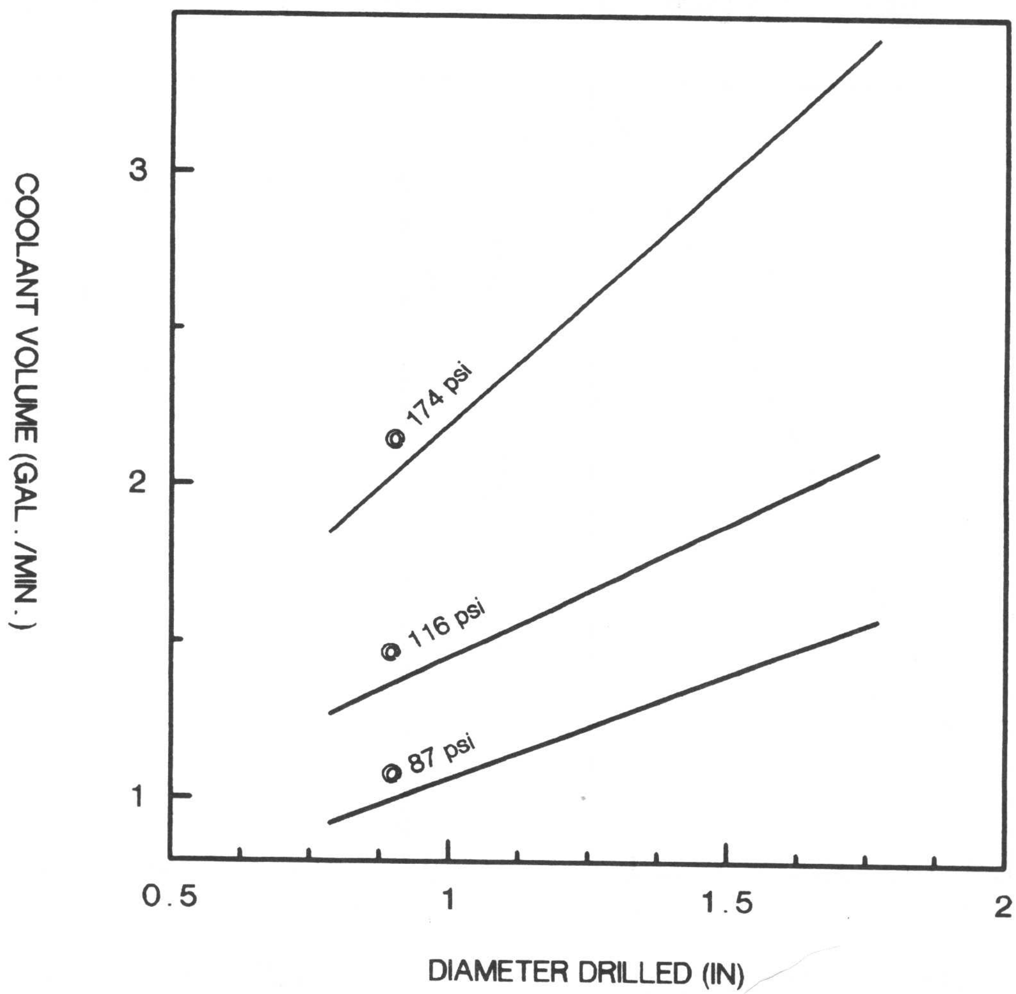




The coolant volume and pressure required to reliably operate an HTS drilling system is greater than the requirements for many indexable drills. This can be attributed to the relative volume of material which must be evacuated from the hole as the tool feeds into the workpiece. HTS tools tend to be of a much larger diameter than catalog standard indexable drills and they are designed to attain much greater hole depths. Hence, the distance a chip travels from the bottom to the top of a hole is much greater, therefore additional coolant pressure and volume is required. For these reasons, whenever possible, HTS should be used in a horizontal drilling mode.

The coolant volume and pressure required to utilize an HTS-C drill is as follows:

HTS-C COOLANT VOLUME VS. DIAMETER DRILLED



The above HTS-C coolant requirements are much lower when compared to HTS in terms of volume required, but the pressure requirements are higher due to the relatively small coolant passages on HTS-C pilot drills. In addition, chips travel a maximum distance of five diameters with HTS-C compared to eight with HTS and therefore the coolant volume requirements are not as dramatic. Increased coolant pressure and volume greatly enhance the tool life generated during deep hole drilling, especially on the pilot drill. In those instances where a hole depth of one diameter is required, external flood coolant is normally sufficient for both HTS and HTS-C.

FIELD TEST RESULTS

The following tests were completed using HTS and HTS-C modular indexable drills. These results have all led to purchase orders by each customer:

Drill Diameter	RPM	Cutting Speed SFPM	Feed Rate IPR	IPM	Hole Depth	Material	# of Holes
HTS 3.50"	207	190	.005	1.03	32.5"	300M steel (32-35 Rc)	2

This application was done previously with a HSS spade drill which required 270 minutes to complete a part, while the HTS required just 32 minutes.

Drill Diameter	RPM	Cutting Speed SFPM	Feed Rate IPR	IPM	Hole Depth	Material	# of Holes
HTS 6.00"	160	250	.006	.96	21.0"	4340 steel (35-38 Rc)	unknown

This application was also done previously with a HSS spade drill which required 210 minutes to complete a part, while HTS required just 22 minutes.

Drill Diameter	RPM	Cutting Speed SFPM	Feed Rate IPR	IPM	Hole Depth	Material	# of Holes
HTS-C .87"	1100	250	.003	3.46	3.66"	4150 steel (32 Rc)	unknown

This tool reduced the cutting time by 38% over the HSS drill previously used on this operation. The cycle time was reduced by over 50% because retracting the drill to clear chips four times per cycle was no longer necessary.

Drill Diameter	RPM	Cutting Speed SFPM	Feed Rate IPR	IPM	Hole Depth	Material	# of Holes
HTS-C 1.18"	1270	393	.004	5.08	4.65"	1022 steel	unknown

This tool reduced the cycle time of the operation by 36% vs. the previously used HSS twist drill.

PRODUCT SELECTION

The extensive Hertel drill product offering provides a wide variety of options to satisfy nearly all of a customers high performance drilling needs. These tools are available from .118"- 6.69" in cutting diameter. Depending on the product selected, holes can be drilled from 2.5-8 diameters in depth. However, which tool should you select when you have a choice of two products with the same cutting diameter?

Since productivity is essential in today's competitive world, the product selected, first, should always be the most efficient tool available. Therefore, when the choice is between an SE drill and Drill-Fix for a hole in the .63"-.79" diameter range on a steel or cast iron part, select the SE drill. Why, because the SE drill provides more predictable and consistent tool life on these materials at higher penetration rates. Just compare the recommended feed rates for a 1018 steel with both tools. With a .63" diameter SE drill, the average catalog recommendation for this material is about 36 IPM while for Drill-Fix it would be about 16 IPM. The SE drill feed rate is calculated at 394 SFPM while the Drill-Fix was at 750 SFPM. Where do you think the best tool life will occur?

When a shallow hole of 2.5 diameters in depth or less is being considered, and the diameter range is from .79"- 1.77", what tool should you select? Since Drill-Fix and HTS-C are the only two standard products available in this size range, the choice is easy. Choose Drill-Fix. Compare recommended operating parameters for similar workpiece materials and tool diameters and you will find that Drill-Fix has the capability to perform at higher operating speeds and feeds. HTS-C and HTS both utilize HSS pilot drills which limit their ultimate cutting speeds and, thus, feed rates. Since Drill-Fix uses coated and uncoated carbide inserts exclusively, this tool will always run at a higher cutting speed and penetration rate than either HTS-C or HTS for a diameter and hole depth within its operating range.

In summary, there are several Hertel drill product choices for a customer. The ultimate selection of a particular tool isn't always an issue of performance. In some cases, the customer might want to avoid regrinding an SE drill even when it demonstrates excellent performance. Or, larger companies often want to limit the number of items they inventory and, hence, they standardize on the most versatile tools, not the most productive. But when performance is the main criteria, don't select the tool your most comfortable or familiar with, choose the most productive tool for the application.

HARDNESS COMPARISON TABLE

Tensile Strength (PSI)	(N/mm)	BHN	Rb	Rc
60,200	415	124	71.2	-
65,300	450	133	75.0	-
69,600	480	143	78.7	-
74,000	510	152	81.7	-
79,000	545	162	85.0	-
83,400	575	171	87.1	-
88,500	610	181	89.5	-
92,800	640	190	91.5	-
97,900	675	199	93.5	-
102,200	705	209	95.0	-
107,300	740	219	96.7	-
111,700	770	228	98.1	-
116,600	800	238	115.1	-
118,900	820	242		23.1
123,300	850	252		24.8
127,600	880	261		26.4
130,500	900	266		27.1
134,900	930	276		28.5
137,800	950	280		29.2
144,300	995	295		31.0
149,400	1030	304		32.2
153,700	1060	314		33.3
158,800	1095	323		34.4
163,100	1125	333		35.5
167,500	1155	342		36.6

Tensile Strength (PSI)	(N/mm)	BHN	Rc
172,600	1190	352	37.7
176,900	1220	361	38.8
182,000	1255	371	39.8
187,000	1290	380	40.8
191,400	1320	390	41.8
195,800	1350	399	42.7
200,800	1385	409	43.6
205,900	1420	418	44.5
211,000	1455	428	45.3
215,300	1485	437	46.1
220,400	1520	447	46.9
225,500	1555	456	47.7
231,300	1595	466	48.4
236,400	1630	475	49.1
241,400	1665	485	49.8
246,500	1700	494	50.5
252,300	1740	504	51.1
257,400	1775	513	51.7
262,500	1810	523	52.3
267,500	1845	532	53.0
272,500	1880	542	53.6
278,400	1920	551	54.1
283,500	1955	561	54.7
289,300	1995	570	55.2
294,400	2030	580	55.7

INDEX OF TERMS

REVIEW QUESTIONS

1. Why are SE, Drill-Fix, HTS and HTS-C drills used without bushings?

2. What target groups of materials was the SE drill designed to drill?

3. Why does the SE drill exhibit superior performance in terms of penetration rates and tool life when compared to HSS drills?

4. When evaluating a potential drilling opportunity for either the HTS or HTS-C modular drilling systems, what single event must occur relative to the pilot drill for these tools to function?

5. What series of materials can the HTS and HTS-C series drills machine?

6. What type of drilled hole tolerance should you expect with an SE and Drill-Fix drill?

7. If a .5" diameter SE drill is to be used to drill a 1" deep hole on 4140 steel, what speed and feed would you run?

8. How does coolant contribute to the success of a drilling operation?

9. What safety precautions should be taken when using HTS-C, HTS and Drill-Fix drills?

10. Explain how the diameter of an HTS and HTS-C drill can be altered?

11. Can you alter the cutting diameter of a Drill-Fix drill in a rotating application? How?

12. What are the maximum recommended hole depths for the SE HTS, HTS-C and Drill-Fix drills?

13. If you were asked to drill 17mm diameter hole, 40mm deep in 1018 steel, using a 15 horsepower machine, what tool would you select?

14. When drilling inclined surfaces with the SE drill, should the operating parameters in terms of speeds and feeds remain the same as if the surface was flat?

15. What operations can Drill-Fix perform in addition to drilling from solid?

16. What direction should a Drill-Fix be offset on a lathe to increase the drilled hole size?

17. Do the pilot drills on HTS and HTS-C modular drills have constant length settings relative their O.D. inserts?

18. Give three reasons why an SE drill might break in a drilling application?

19. What speed and feed would you use to drill a 3.75" dia. hole 15" deep in a 4140 steel workpiece using a 40 horsepower machine?

20. How much horsepower is required to drill the holes in question number 7 and 19?

NOTES

NOTES

TRAINING MANUAL

GLOSSARY OF TERMS

finding better ways KENNAMETAL®

GLOSSARY OF TERMS

Abrasion Resistance - The ability of a material to withstand a change in dimensions due to a rubbing action with another material. As tested for carbide, a steel wheel with an aluminum-oxide slurry is rubbed over the test piece. (Cemented Carbide Producers Assn. Physical Test P-112, "Abrasive Wear Resistance.")

Abrasive Wear - The wear that occurs on a tool in use due to rubbing action in machining. (See Flank wear)

Age Hardening - Hardening by aging, usually after rapid cooling or cold working.

Air Chuck - Air operated chucking device using standard double angle collets.

Allowance - An intentional difference in dimensions of mating parts. It is the minimum clearance (positive allowance) or maximum clearance (negative allowance) between such parts.

Alloying Element - An element added to a metal to effect changes in properties and which remains within the metal.

Anvil (Seat, Shim) - A removable part of a toolholder designed to provide support for the cutting insert. (See Insert package)

Annealing - Heating to and holding at a suitable temperature and then cooling at a suitable rate for such purposes as reducing hardness, improving machinability, facilitating cold working, producing a desired microstructure, or obtaining desired mechanical, physical or other properties. When applicable, the following more specific terms should be used:

black annealing	full annealing	process annealing
blue annealing	graphitizing	quench annealing
box annealing	intermediate annealing	recrystallization annealing
bright annealing	isothermal annealing	spherodizing
flame annealing	malleablizing	

When applied to ferrous alloys, the term "annealing", without qualification, implies full annealing. When applied to nonferrous alloys, the term "annealing" implies a heat treatment designed to soften a cold worked structure by recrystallization or subsequent grain growth or to soften an age hardened alloy by causing a nearly complete precipitation of the second phase in relatively coarse form. Any process of annealing will usually reduce stresses, but if the treatment is applied for the sole purpose of such relief, it should be designated as "stress relieving."

Apparent Density - The weight of a unit volume of powder generally in the unpacked or untamped condition. This is usually expressed in terms of grams per cubic centimeter, determined by a specified measure such as ASTM Standard B212-48(1970) or expressed in terms of grams per cubic inch as in ASTM Standard B329-70. (See also Scott Density)

Arbor - A device to mount in or on the spindle of a machine tool and which is designed to carry and drive a peripheral milling cutter. (See also Stub-Arbor)

Armor-Coat - Cemented carbide which has been coated with materials designed to give improved machining or wear performance. ARMOR-COAT is the trade-marked name for accomplishing this by a proven patent-pending method.

Attack Angle (Lead Angle, Side-Cutting -Edge Angle). - Term preferred by some to avoid confusion over the meaning of Side-Cutting-Edge Angle. (See Lead Angle)

Austenite - A solution of one or more elements in face-centered cubic iron. Unless otherwise designated (such as nickel austenite), the solute is generally assumed to be carbon.

Automatic Press - A compacting press, either mechanical or hydraulic, capable of repeat operation without stopping, often referred to as a "pill press".

Axial Rake - This is measured between the peripheral cutting edge and the axis of the cutter when looking radially at the point of intersection. For a cutter with helical teeth, the axial rake angle equals the helix angle.

Back Rake (Top Rake) - The angle of inclination of the face the tool away from the end cutting edge. It is measured in a plane perpendicular to the base of the tool and parallel to the side cutting edge. (See Rake)

Bar (Boring Bar) - A toolholder specifically designed to support cutting tools in a boring operation. Deep-hole boring often requires a long, round (or “bar-shaped”) toolholder that is supported on one end.

Base - That surface of the shank which bears against the support and takes the tangential pressure of the cut.

Basic Oxygen Process - A rapid method of making steel utilizing the exothermic heat generated by high purity oxygen reacting with carbon and silicon in the pig iron charge.

Bending Moment - The calculated force that causes movement of a beam held at one point and under a load at another point. Mathematically, it is the algebraic sum of the couples or moments of the external forces or both to the left or to the right of any station on a member subjected to bending by couples of transverse forces or both. Thus, a boring bar is subject to bending moment. The amount it bends in elastic behavior (prior to permanent deformation) depends upon the length of the bar, the diameter, and the modulus of elasticity of the material. The modulus of elasticity is essentially the same for steels regardless of hardness but is much higher for cemented tungsten carbide, for example, and much lower for aluminum. The larger the bar diameter or the higher the modulus of elasticity, the less bending will occur under a given load or bending moment. Increased bar length increases the bending moment. (See Modulus of Elasticity)

Bend Radius - The inside radius of a bent section.

Bend Test - A test for determining relative ductility of metal that is to be formed, usually sheet, strip, plate or wire and for determining soundness and toughness of metal. The specimen is usually bent over a specified diameter through a specified angle for a specified number of cycles.

Billet - A solid semifinished round or square product that has been hot worked by forging, rolling or extrusion. An iron or steel billet has a minimum width or thickness of 1 1/2 inches and the cross-sectional area varies from 2 1/4 to 36 square inches. For nonferrous metals, it may also be a casting suitable for finished or semifinished, rolling or for extrusion.

Binder - The metallic constituent in cemented tungsten carbide or cemented titanium carbide which holds the carbide grains together. The binder is generally cobalt for tungsten carbide and nickel or titanium carbide or may be a combination of cobalt, nickel and iron. Other binder metals are sometimes used for specific applications.

Birds Nest Chip - Long, stringy continuous chip which randomly wraps around itself.

Blank - A sintered piece of generally regular rectangular or cylindrical shape and usually in the unground condition.

Blend - A mixture of two or more components intended to be as homogeneous as possible. A blend may be a mixture of immiscible (non-soluble) liquids or may be a mixture of two or more powdered solids. In today's technology, the various carbide powder and binder components are generally said to be "blended" in a mill. A blend may also be homogenization of each component to make it as uniform as possible throughout. For example, WC powder may be blended into a large batch to obtain a sample or to insure uniform production.

Boring - A machining process in which internal diameters are made in true relation to the centerline of the spindle. It is most commonly used for enlarging or finishing holes or other circular contours.

Braze - To join two materials with a third lower melting point material, the braze alloy. This is stronger than a soft solder joint. For carbide "silver solder" is generally used of the American Welding Society designation BAg-3. This is a braze as contrasted to a lead-tin soft solder.

Breakage - Catastrophic failure of an insert - large section separates.

Brinnel Hardness Test (Bhn) - System for measuring hardness of metal by forcing a hard steel or carbide ball of specified diameter into it under a specified load. The result is expressed as the Brinell hardness number, which is the value obtained by dividing the applied load in kilograms by the surface area of the resulting impression in square millimeters.

Broach - A multiple tooth cutting tool with progressively larger teeth which is designed to finish a surface in a single stroke. A broach may be used to produce an shaped surface, a groove or hole.

Buildup - The welding of chips to the cutting tool. It is a major cause of surface roughness. (See Built-Up Edge.)

Built-Up Edge- An adhering deposit of work material on the tool face (rake face) adjacent to the cutting edge. (See Buildup)

Bulk Density - Generally synonymous with Apparent Density (U.S.) and Loading Weight (British). Bulk density is the term most often used in the chemical industry and may refer to the semi-tamped condition used to ship chemicals in the powder form.

Bullet - Green slab or block cold compacted or hydrostatically pressed and often sintered for slicing into smaller sizes.

Burning - (1) Permanently damaging a metal or alloy by heating to cause either incipient melting or intergranular oxidation; (2) In grinding, getting the work hot enough to cause discoloration or to change the microstructure by tempering or hardening.

Burnisher - Non-cutting teeth, generally with spherical surfaces, which compress, cold work or burnish the overall work surface to impart a finish.

Burnishing - A method of finishing by means of compressing or cold working the work surface with carbide rollers called burnishing rolls or burnishers.

"C" Classification - Carbide grade applications system established as an industry standard way to group all carbide grades. This system does not, however, imply equal performance for all grades in any given category. Rather, it only identifies the suitable applications for a given insert.

Carbide Bar - Boring bar with tungsten carbide body - reduces chatter.

Carbon - A nonmetallic element having an atomic number of .6 and atomic weight of 12.01115. it is commonly seen as carbon black, lamp black or graphite. It has a density of 2.25 g/cm^3 as graphite. Carbon sublimes above 3,500^{o}C and has a boiling point of 4,827^{o}C. It combines readily at high temperatures with many metallic elements to form the various carbides such as WC (tungsten carbide), TiC (titanium carbide), THC (tantalum carbide).

Carbon Deficiency - Carbon content less than that needed to produce a sintered carbide consisting of WC plus cobalt and solid solution carbides. A sintered carbide containing eta phase.

Carbon Excess - Carbon content more than that needed to produce a sintered carbide consisting of WC plus cobalt and solid solution carbides. A sintered carbide containing carbon porosity ("C" type porosity).

Carbon Porosity - An acicular type of microscopic porosity caused by very small pores of excess carbon in the sintered carbide.

Carbon Steel - Steel containing carbon or to about 2% and only residual quantities of other elements except those added for deoxidization, with silicon usually limited to 0.60% and manganese to about 1.65%. Also termed "ordinary steel", "straight carbon steel", "plain carbon steel".

Carbonize - The process of chemically combining carbon and a metal to form a carbide generally carried out by mixing (blending) finely divided carbon powder and a metal powder and heating the mixture in a protective atmosphere or vacuum to form the metallic carbide such as WIC.

Case - In a ferrous alloy, the outer portion that has been made harder than the inner portion, or core, by case hardening.

Case Hardening - Hardening a ferrous alloy so that the outer portion, or case, is made substantially harder than the inner portion, or core. Typical processes used for case hardening are carburizing, cyaniding, carbonitriding, nitriding, induction hardening and flame hardening.

Cemented Carbide - Most often tungsten carbide (with or without additives of titanium carbide and/or tantalum carbide) with cobalt binder or titanium carbide with nickel molybdenum carbide binder. This term, however, refers to all the sintered carbide materials in general.

Center Line - Imaginary plane which bisects the workpiece and is at a 90^{o} angle to the headstock or table.

Centrifugal Casting - A casting made by pouring metal into a mold that is rotated or revolved.

Cermet - A material generally composed of a nonmetal compound or a metalloid and a metal. From the word ceramic and metal combined. Cemented carbide is a cermet, but even tungsten combined with copper is referred to as a cermet.

Chamfer - A bevel on the cutting edge of a cutting tool for the purpose of increasing its strength. The angle is measured from the cutting face downward and will generally vary from 25 degree to 45 degree.

Charpy Test - A pendulum-type single-blow impact test in which a specimen, usually notched, is supported at both ends as a simple beam and broken by a falling pendulum. The energy absorbed, as determined by the subsequent rise of the pendulum, is a measure of impact strength or notch toughness.

Chaser - Replaceable cutting edge which is inserted into threading dies.

Chattering - Vibration between tool and work sufficient to cause an irregularity in the tool marks on the finished surface.

Chip - A fragment of the work material which has been separated in the machining operation. In general the ideal chip is shaped like a figure 6 or a figure 9.

Chipbreaker - A groove or irregularity in the face of a tool, or a separate piece fastened to the tool or toolholder, to cause the chip to break into short sections, or curl.

Chipbreaker Insert - An insert with a built-in chipbreaker. This generally consists of a groove around the top face of the insert near the periphery.

Chip Clearance - In milling, the groove or space provided in the cutter body to allow chips to be formed by the inserts.

Chip Control - Procedures used to determine the shape and direction of the metal being removed from the workpiece.

Chipping - The breakdown of cutting edges by loss of fragments broken away during the cutting action.
Chucker - Machine normally used to cut a part whose diameter is larger than length.

Clamp - The clamp locks the insert in the pocket.

Clearance - The angle below or behind the cutting edge, which allows the cutting-edge to be forced into the work. Without clearance, the tool will not cut. It is also the term used for secondary relief in some cases. (See Relief)

Climb Milling - In climb milling, the milling cutter attempts to "climb" the workpiece. Chips are cut to maximum thickness at initial engagement of cutter teeth with work and decrease to zero thickness at the end of engagement. (See also Conventional Milling)

Coarse Powder - A powder that has a particle size above about 5 to 10 microns. Often a powder that has a gritty feel, or where the particles are large enough to be seen as individual particles with the naked eye. Salt is coarse powder and flour is fine powder.

Cobalt (Co) - A relatively soft, ductile, magnetic metal element having a density of 8.85 g/cm^3 and a melting point of 1,495^{o}C (2,723^{o}F). This metal is the metal cement (or binder) most often used for cement tungsten carbide compositions.

Cobalt Content - The amount of cobalt in a cement tungsten carbide cobalt material with or without titanium carbide and tantalum carbide added. The cobalt content is generally expressed as a weight percent of all of the constituents present. It may, at times, however, be expressed in terms of volume percent and can be estimated by examining the binder constituent in the metallographic structure.

Coercive Force - The magnetizing force that must be applied in the direction opposite to that of the previous magnetizing force in order to remove residual magnetism; thus, a measure of the magnetic retention of magnetic materials. Values given in Oersteds of cemented carbides; the coercive force indicates the grain size, binder content and relative carbon content or carbon deficiency. Grain size decrease increases coercive force. Cobalt decrease increases coercive force. Carbon decrease from theoretical or carbon deficiency (eta phase) increases coercive force. It is difficult to analyze results because the same reading may indicate the interaction of different variables. Coercive force is best used for detecting carbon deficiency.

Cold Press - Powder compacted under pressure of ambient temperature to form a green shaped piece.

Cold Work - Permanent strain produced by an external force in a metal below its recrystallization temperature.

Collet Chuck - A means of gripping a part of its O.D. for drilling reaming, etc.

Columbium Carbide (CbC) - A compound of equal parts by atomic weight of Columbium and carbide. Columbium carbide (CbC) and niobium carbide (NC) are identical. The International Chemical Society accepted the term niobium for this element rather than Colombian, but the use of Columbium persists in the metals industry in the U.S. The carbon content of Columbium carbide is 11.46 weight percent and Columbium content is 88.54 weight percent. Columbium carbide has a melting point of 3,480^{o}C (6,300^{o}F) and a Vickers microhardness of 2400 kg/mm^2 and a density of 7.80g/cm^3. This compound, in the form of a fine powder, is used as an additive to cobalt-cemented tungsten carbide in steel cutting grades for similar use and in conjunction with tantalum carbide.

Compact - An object produced by the compression of carbide powders, generally well confined in a die. The resultant piece from cold pressing.

Compacting - The forming of an object from powder and the compression of the powder, generally while contained in a die, with or without the inclusion of temporary or permanent binder constituents.

Compression - The ability of a tap chuck to compress when the machine is feeding the tap into the work but the tap has not yet started cutting

Compressive Strength - Ultimate or breaking strength in pounds per square inch (psi) or kilograms per square millimeter (kg/mm^2) when subjected to compression.

Crater - A grooved area or depression caused by chips rubbing-away or eroding a groove or well in the top (the rake face) of the insert behind the cutting edge of a tool.

Cratering - The action whereby an insert face gets a depression in it from being eroded by chip contact. (See Crater)

Critical Point - (1) The temperature of pressure at which a change in crystal structure, phase or physical properties occurs. Same as transformation temperature; (2) In an equilibrium diagram, that specific value of composition, temperature and pressure, or combinations thereof, at which the phases of a heterogeneous system are ion equilibrium.

Cutting Edge - That part of the face edge along which the chip is separated from the work. The cutting edge consists of the side cutting edge, the nose, and the end cutting edge.

Cutting Speed - In turning, the peripheral velocity of the workpiece at the cutting radius. It is measured in surface feet per minute, sfm.

Decarburization - Loss of carbon from surface of a ferrous alloy as a result of heating in a medium that reacts with carbon at the surface.

Deformation - The permanent change in the shape of a cutting tool due to cutting forces and temperature. This generally occurs in high-speed or heavy machining.

Deoxidizing - (1) The removal of oxygen from molten metals by use of suitable deoxidizes; (2) Sometimes refers to the removal of undesirable elements other than oxygen by the introduction of elements or compounds that readily react with them; (3) In metal finishing, the removal of oxide films from metal surfaces by chemical or electrochemical reaction.

Depth Of Cut - The distance between the bottom of the cut and the uncut surface of the work, measured in a direction at right angles to the machined surface of the work. This is the difference in height between the machined and unmachined surfaces.

Dial-Set - A system of adjusting boring tools within .0001/inch increments for two point cutters.

Diaphragm Chuck - Most accurate chuck with no moving parts. Chucking done by the spring pressure of a diaphragm.

Double Angle - Two angles in the same plane that gives the effect of one long angle.

Double Pin Floating Holder - A tool for holding reamers, allowing angular float only. Cannot be used in rotating applications.

Drawing - (1) Forming recessed parts by forcing the plastic flow of metal in dies; (2) Reducing the cross-section of wire or tubing by pulling it through a die. A misnomer from tempering.

Ductility - The ability of a material to deform plaitically without fracturing, being measured by elongation or reduction of area in a tinsile test, by height of cupping in an Ericksen test or by other means.

Edge Preparation - A conditioning of the cutting edge, such as honing or chamfering. (See Chamfer, Honing)

Edge Wear - Erosion of the section of the insert passed over by the chip - caused by adhesion of diffusion and abrasion.

End-Cutting-Edge Angle - The angle between the cutting edge on the end of the tool and a line perpendicular to the side edge of the straight portion of the tool shank.

End Relief Angle - Angle between the portion of the end flank immediately below the cutting edge and a line drawn through that cutting edge at right angles to the base of the toolholder. (See Clearance, Relief)

Engine Lathe - A floor mounted machine on which work is rotated about a horizontal axis and shaped by a cutting tool.

Entrance Angle - The angle that the side-cutting edge of a tool makes with the machined surface of the work, measured on the cutting-edge side of the tool point.

Eta Phase - The intermetallic compound of cobalt, tungsten and carbon (CO_4W_2C) formed because of carbon deficiency in the sintered carbide.

Expanding Collet - A device used to hold in the I.D. of a part and has the outside configuration to be used in a collet chuck.

Extrude - In cemented carbide, to make rod or tubing. The powder, in a paste-like consistency due to additives which are later driven off by heat, is cold or warm forced under pressure through a nozzle.

Extrusion - Conversion of a billet into lengths of uniform cross section by forcing the plastic metal through a die orifice of the desired cross-section outline. In "direct extrusion," the die and ram are at opposite ends of the billet, and the product and ram travel in the same direction. In "indirect extrusion" (rare), the die is at the ram end of the billet and the product travels through and in the opposite direction to the hollow ram. A "stepped extrusion" is a single product with one or more abrupt cross-section changes and is obtained by interrupting the extrusion by die changes. "Impact extrusion" (cold extrusion) is the process or result product of a punch striking an unheated slug in a confining die. The metal flow may be either between the punch and die or through another opening. Also Hooker process, which used a pierced slug. "Hot extrusion" is similar to cold extrusion except that a preheated slug is used and the pressure application is slower.

F.P.M. - Feet per minute.

Fail Safe Air Cylinder - Air cylinder used to actuate chucks, mandrells, etc., with built-in safety. Air Cylinder traps air pressure inside with ball check valves in the event of system pressure loss.

Fatigue - The phenomenon leading to fracture under repeated or fluctuating stress having a maximum value less than the tensile strength of the material. Fatigue fractures are progressive, beginning as minute cracks that grow under the action of the fluctuating stress.

Face (Rake Face) - That surface of the cutting tool on which the chip impinges as it is separated from the work. Also, to machine the flat or end surface of the workpiece, such as facing a surface of a bar before or after turning.

Face Mill - A milling cutter which used its face to generate the surface of the work, although the outside diameter or bevel cutting edge of the cutter removes most of the stock. The cutter is driven by a spindle on an axis perpendicular to the surface being milled. Face mills may be mounted directly upon the machine spindle or by means of a stub arbor and are driven by a keyway across the back of the cutter. (See also End Mill, Shell End Mill)

Facing - Machining along the centerline towards the center of the end of the workpiece.

Feed - In turning, the distance moved by the tool into the work for each revolution of the work. It is measured in inches per revolution, IPR.

Figure 6 Chip - Metal removed from the workpiece which bonds back towards the workpiece and breaks - shaped like a handwritten 6.

Fine Pitch Cutter - A milling cutter with more than the standard number of cutting edges. Usually a cutter having a number of cutting edges three to four times the diameter of the cutter in inches. Example: A 6-inch cutter with 18 to 24 cutting edges.

Finish Cut - Final pass of the cutting tool - light feed and depth of cut at high surface speed.

Flange - The projecting annular rim around a cylinder, used for strengthening, fastening or positioning. A circular metal plate that drives a grinding wheel.

Flaring - Forming an outward acute-angle flange on a tubular part. Forming a flange by using the head of a hydraulic press.

Flank - That surface adjacent to the cutting edge and below it when the tool is in a horizontal position for turning.

Flank Wear (Abrasive Wear) - The wear that occurs along the flank of a tool, below and immediately adjacent to the cutting edge, while cutting. This wear reduces the clearance angle of the tool until failure finally occurs. (See Wear Land)

Flat - The straight portion of the end cutting edge at zero degree angle, intended to eliminate feed marks and produce a smooth machined surface. (Used most often in milling).

Flute - Helical or straight groove cut or formed in the body of the tool to provide cutting lips, to permit removal of chips, and to allow cutting fluid to reach the cutting edge. (See also Chip Clearance)

Force - Any cause which tends to produce or mock motion. It is usually measured in pounds. Force has the characteristics of direction, place of application and magnitude.

Forging - Squeezing red hot steel between dies to strengthen it.

Form Tool - A tool ground to a shape so that the cutting edge forms a desired contour, the matrix of which is formed on the work.

Full Annealing - Annealing a ferrous alloy by austenitizing and then cooling slowly through the transformation range. The austenitizing temperature for hypoeutectoid steel is usually Ac3; and for hypereutectoid steel, usually between Ac1 and Ac...

Full Floating Holder - A tool for holding reamers, allowing them to float for use with drill bushings or when reamers must follow previous drill holes. Holder floats radially and angularly.

Full Side Mill - A peripheral milling cutter having the keyway in the mounting hole and designed to cut on the periphery and both sides. (See also Half Side Mill, Side Mill, Peripheral Mill, Staggered Tooth Milling Cutter)

Galling - Developing a condition on the rubbing surface of one or both mating parts where excessive friction between high spots results in localized welding with subsequent spalling and a further roughening of the surface.

Gear Chuck - A chuck specifically designed to hold gears at the pitch line.

Grade - A designation given to a composition for a particular cemented carbide. The composition may be published or may be proprietary.

Grain Size - The size of a grain (a particle imbedded in the cementing binder) as seen in a flat, plain polished metallographic section. The grain size of cemented tungsten carbide varies from less than one micron to more than ten microns (1 micron equals 0.00003937 inches). The grain size is generally measured at a magnification of 1500 diameters. The method of measuring is found in "Recommended Practice for Evaluating Apparent Grain Size and Distribution of Cemented Tungsten Carbide" ASTM designation B390-64 (1970).

Granulated Powder - Powder that has been completely formulated with its binder and lubricant, such as paraffin added, and then formed into coarse granules of material. A granulated powder looks like sand or granulated sugar. The granules can be readily crushed between the fingers and are formed as irregularly shaped coarse particles only to facilitate die fill and flow in the pressing operation.

Green - Not sintered. As compacted or as cold pressed before sintering. May also refer to a pre-sintered piece.

Grinding - Metal removal operation using an abrasive wheel - usually to improve surface finish or alter materials size.

Grit - Abrasive particle. Particles of coarse WC, generally coarser than 325 mesh (44 microns or larger). Also crushed sintered cemented tungsten carbide or crushed "cast carbide", all generally coarser than 325 mesh. Cast carbide is predominantly W_2C.

Grooving - Operation to cut narrow channel in rotating workpiece.

Ground - Diamond wheel abrasive finished after sintering.

Half Side Mill - A peripheral milling cutter having a keyway in the mounting hole and designed to cut on the periphery and one side only. It is important to know if it is right or left-handed. Looking at the side which cuts, clockwise rotation indicates a left-hand cutter; counterclockwise rotation indicates a right-hand cutter. (See also Full Side Mill, Peripheral Mill, Side Mill)

Hard Drawn - Temper of copper or copper alloy tubing drawn in excess of 25% reduction in area.

Hardness - Ability to withstand deformation generally measured by the depth or area of penetration under a fixed load using a diamond indenter. On cemented carbide, this is measured on the Rockwell A Scale in the U.S. and on the Vickers Scale in Great Britain, Europe and most other countries. On steel and other ferrous metals, Rockwell C or Brinell (BHN) is generally used. For nonferrous metals, Rockwell B or Rockwell as well as Brinell is used.

Hardenability - In a ferrous alloy, the property that determines the depth and distribution of hardness induced by quenching..

Heat Check - Many small cracks in the surface caused by grinding too rapidly, without proper coolant, or with too much pressure during grinding.

Heat Cracking - Thin cracks in carbide inserts caused by rapid heating and cooling.

Heat of Deformation - Heat caused by the slipping and rubbing of the grains in the workpiece material.

Heat Treating - Process used to improve the hardness of metal by applying heat through various methods. Heating and cooling a solid metal or alloy in such a way as to obtain desired conditions or properties. Heating for the sole purpose of hot working is excluded from the meaning of this definition.

High Speed Steel - Steel containing about 1% carbon plus the alloys of tungsten and manganese.

Holder Style - Design based on shank offset plus side or end cutting edge angle.

Honed - Abrasively finished, generally by rubbing, using a wet compound and abrasives.

Honing - The process of rounding or blunting the cutting edge with abrasives for increased edge strength. It may be done by hand or by machine. Standard hone specifications: A, light hone, 0.001 to less than 0.003 in. radius: B, medium hone, 0.003 to 0.005 in., and C (or H), heavy, 0.005 to 0.007 in. (See Edge preparation).

Hot Press - A method of producing a solid body from powder by simultaneously applying heat and pressure. In carbide, generally done in graphite dies in temperatures up to those used in sintering.

Hour Glass - Term used to describe a narrowing in the middle of a cylinder, generally caused by compacting the powder at both ends so that because of die wall friction, the powder in the center is compacted slightly less than at the ends and therefore shrinks more.

Hydraulic Powder - Fine powder that has been formulated from the carbide and the binder constituency and generally has the paraffin lubricant added in a ready-to-press condition. Hydraulic powder has the consistency of talcum powder and is soft to the touch. It is used for filling large molds where automatic high speed pressing is not involved, or for hydrostatic pressing.

Hydraulic Press - A compacting press that employs liquid as motive force.

Hydrogen (H_2) Sinter - See Sinter.

Hydrostatic Press - A device for compacting employing a closed, high-strength vessel where metal powder is contained in a rubber or plastic closed envelope can submerged in a liquid and then brought up to high pressures of as much as 30,000 to 50,000 psi for directional compacting. (See also Isostatic Press)

I.C. - Inscribed circle - a circle which touches all sides of an insert - used to establish size. Measurements are in fractions of an inch and describe the diameter of the circle.

ID - Inside diameter.

Inclusions - Nonmetallic materials in a solid metallic matrix.

Indexing Chuck - A chuck used for holding parts with intersecting axes, producing finished parts with one chucking.

Insert Geometry - Physical characteristics of insert.

Insert Grades - Chemical properties of inserts.

Internal Machining - (Boring) - enlarging an existing hole to a predetermined shape.

I.P.M. - Inches per minute.

I.P.R. - Inches per revolution. (See also Feed)

Included Angle - A measurement of the total angle within the interior of a piece. The angle between any two intersecting lines or surfaces. The corner angle of an insert.

Indexable Insert - A carbide cutting tool which has several cutting edges and is used in milling cutters angle point tooling. An indexable insert is designed so that all cutting edges may be used to the limit of wear before the tool is discarded or reground.

Inscribed Circle(IC) - The circle that can be constructed internal to any closed figure or shape so that all sides of the figure are tangent to the circle. The inscribed circle is most often used to describe the dimensions of a triangle, pentagon, hexagon, octagon, or trigon. The IC of a square or parallelogram is equal to the perpendicular distance between opposite sides; IC of a round insert is the same as the insert diameter.

Insert - The cutting tip, made of hard material, that is mechanically affixed to the toolholder for use. It can generally be indexed to present more than one cutting edge.

Insert Package - The complete assembly that fits into the insert pocket of a toolholder. This includes such components as a seat, setscrew, insert, chipbreaker, clamp and clamp screw.

Insert Pocket - The space that has been machined out of a toolholder to receive the insert package (seat, chipbreaker, etc.)

Iron Cutting Grade - Tungsten carbide grade whose binder is primarily cobalt.

Isostatic Press - A device for compacting or bonding that employs a gas or liquid as motive force for all directional compacting. Hydrostatic pressing is a form of isostatic pressing. Generally, hydrostatic refers to the use of liquid at room temperature and isostatic is used for the device employing gas, generally at elevated temperatures. Hydrostatic is most commonly used as an initial pressing operation in the carbide industry, but hot isostatic pressing is now also used for the removal of porosity after sintering.

Job Shop - Machine shop which produces small quantities of a wide variety of parts.

Lamination - A discontinuity, crack or slit parallel to the plane of pressing and generally caused by improper compacting of the piece. Metal defects with separation or weakness generally aligned parallel to the workload surface of the metal may be the result of pipe, blisters, seams, inclusions or segregation elongated and made directional by working Lamination defects may also occur in metal-powder compacts.

Lapping - An abrasive finishing process involving rubbing with a wet compound. Commonly used to achieve a high degree of flatness on inserts.

Lead Angle (Side-Cutting Edge Angle, Attack Angle) - The angle between the side cutting edge and the projected side of the tool shank or holder, which leads the tool into the work. (See Attack Angle)

Left Hand Holder - Toolholder which, when viewed from the front, has the cutting edges on the left side of the holder; normally cuts from left to right.

Lock - A device used to clamp inserts into milling cutter or tool holder.

Lock Pin - This pin positions insert back and down in the pocket of the tool.

Lubricated Powder - Powder to which paraffin, wax or other lubricant has been added so that the powder can be readily compacted together to form a cohesive mass of limited and temporary strength prior to the presinter and sintering operations.

Lubricity - Slipperiness; the property that diminishes friction. Tantalum carbide and titanium carbide are used to produce lubricity in steel cutting grades of tungsten carbide to reduce crater and wear.

Machinability Rating - A numerical value expressing the ease or difficulty of machining a particular workpiece material in comparison with AISI B1112 cold-rolled steel being turned at 108 sfm, which is rated at 100%.

Magnetic Particle Inspection - A non destructive test method of determining both the magnitude and frequency of non-metallic inclusions and discontinuities irferromagnetic materials.

Mandrel - A means of gripping a part in its' I.D. for holding while machining, assembling, etc.

Master Spacer - Manually operated positioning device available in 6" and 10".

Mechanical Press - A compacting press that employs mechanical cams or levers for motive force.

Mesh Size - A size used to define powder by means of the coarsest mesh screen which it will all go through and the finest mesh screen upon which it will be retained. Therefore, mesh size refers to the number of strands per inch in the mesh. The smaller the standard sieve or mesh size number, the larger the particle and the larger the opening. For example, in the cemented carbide industry, mesh sizes as coarse as 12 mesh may be referred to which has an opening of .0661 inches (1680 microns), the smallest mesh size commonly used known as 325/400 mesh which has an opening of .0018 inches (44 microns).

Metal Removal Rate - Volume of metal removal; expressed in cubic inches per minute.

Milling - A method of machining metal which involves multiple cutting edges rotating around an axis with the workpiece being moved past this axis. (See also Milling Cutters)

Milling (Rod or Ball) - A process of mixing and breaking down powder particles in a rotating cylindrical vessel containing either balls of regular shape, crude ball shape or rods. The diameter of the balls or rods is generally about 1/4 inch to 1 inch. Milling may be done dry or wet, and in the latter case may be aqueous solvent.

Milling Cutters - Rotary cutting tools provided with one or more cutting elements called teeth, which intermittently engage the workpiece and removes material by movement of both the workpiece and cutter. (See also Milling, End Mill, Face Mill, etc.)

Modified Standard - This is an item in which the standard is modified by a preforming operation in the green state. Also a standard item on which additional work is done.

Modulus of Elasticity - A measure of stiffness or rigidity. It is a stress divided by the strain it produces. Measured in psi, a higher modulus means a more rigid material such as cemented carbides. Also called Elasticity Modulus and Young's Modulus of Elasticity (YME). (See also Bending Moment)

Molybdenum Carbide (MO_2C) - A compound of two parts by atomic weight of molybdenum and one part carbon. The carbon content of molybdenum carbide is 5.88 weight percent, and the molybdenum content 94.12 weight percent. Molybdenum carbide has a melting point of 2,410^{o}C (4,370^{o}F), a Vickers microhardness of 1950 kg/mm^2, and a density of 9.06 g/cm^3. This compound, in the form of a fine powder, is often used as an additive to cemented titanium carbide composition.

Moment of Force- The force multiplied by the perpendicular distance from the fixed point to the direction of force. The fixed point is called the center of moment, and the perpendicular distance is called the lever arm of the force. Moment of force is measured in foot-pounds or inch-pounds.

N/C Machine - Machine with function controlled by a programmed format on tape or directly from a computer.

Negative Geometry - This style insert does not have clearance from cutting edge; clearance is provided in the pocket of the tool. This style offers twice as many cutting edges as a comparable positive rake insert.

Negative Land (K-Land) - A bevel along the cutting edge causing the rake to become more negative. It may range from 3 degree to 15 degree and is measured off the top face of the insert. (See Chamfer)

Negative Rake - A rake angle that is less keen or more blunt than zero rake. (See Rake)

Neutral Hand Holder - Insert is in the center of the tool holder; can cut either right or left.

Neutral Rake - A rake angle of zero degrees. This angle is perpendicular to the surface of the work and neither positive nor negative.

Nib - A pressed, presintered or fully sintered part, usually cylindrical on the outside, shaped to some geometrical figure on the inside diameter which may be mounted in a case to make a wire die.

Nickel (NI) - A relatively soft, ductile, magnetic metal element having a density of 8.90 g/cm^3 and a melting point of 1,453^oC (2,647^oF). This metal is the metal cement (or binder) most often used for cemented titanium carbide compositions.

Niobium Carbide (NbC) - Niobium is the international accepted name for the metallic element frequently called columbium in the U.S. Niobium is identical to columbium and, therefore, niobium carbide is identical to columbium carbide.

Nitriding - Introducing nitrogen into a solid ferrous alloy by holding at a suitable temperature (below Ac_1 for ferritic steels) in contact with a nitrogenous material, usually ammonia or molten cyanide of appropriate composition. Quenching is not required to produce a hard case.

Normalizing - Heating a ferrous alloy to a suitable temperature above the transformation range and then cooling in air to a temperature substantially below the transformation range.

Nose - The corner angle formed by joining the side and end cutting edges of a tool.

Nose Radius - The radius on the tool between the end and side cutting edges.

Nose Ring - Part inside nosepiece that contacts the collet and acts as a thrust washer.

OD - Outside diameter.

Offset Tool - A tool which has the cutting edge extending beyond the side of, but parallel to, the shape.

Open C Chip - Chip shaped like the letter C - almost perfect.

Open-Hearth Furnace - Reverberatory melting furnace with shallow hearth and low roof. Flame passes over charge on the hearth causing the charge to be heated both by direct flame and by radiation from roof and sidewalls of the furnace. In ferrous industry, furnace is regenerative.

Operating Conditions - Feed, speed and depth of cut (also called cutting parameters).

Oversinter - A condition of sintering a piece of cemented carbide at excessive temperature or for excessive time. Oversintered carbide is characterized by excessive grain growth and consequent low hardness and may result in blisters and an increased quantity of large pores.

Oxygen-Free Copper - Electrolytic copper free from cuprous oxide, produced without the use of residual metallic metalloidal deoxidizers.

Particle - A discreet, small chunk of material. The combination of many particles makes up a material known as powder.

Particle Size - The size expressed in microns for small particles (1 micron equals 0.00003937 inches) or in terms of mesh size for coarse particles. Mesh size is used for particles above 44 microns which is equal to 325 mesh. (See definition of mesh size.) Cemented carbides are generally made from particles having sizes of less than 1 micron up to 20 microns. Grain size is sometimes incorrectly referred to as particle size.

Pelletized Powder - A refined form of granular powder (see definition) in which the coarse agglomerated particle is formed into a crude sphere as to further facilitate flow and die fill. Pellitized powder is often produced in selected mesh size ranges.

Peripheral Milling Cutter - A milling cutter which cuts mainly on the periphery to produce the finished surface. A peripheral mill is usually mounted on an arbor with its axis parallel to the surface being milled. It is driven by means of a key which fits into a keyway along the axis of the arbor and passes through a keyway in the mounting hold of the cutter. (Peripheral mills are also classified as slab mills, slotting cutters, side mills, half side mills, angle cutters and form cutters.)

Phase - A metallographic constituent observed in the microstructure of a metal or alloy. In the case of cemented tungsten carbides, phases would be tungsten carbide grains, cobalt binder, tantalum carbide phase, solution phase of tungsten carbide, titanium carbide or solution phase of tungsten carbide, titanium carbide, tantalum carbide.

Physical Properties - The properties, other than mechanical properties, that pertain to the physics of a material; for example, density, electrical conductivity, heat conductivity, thermal expansion.

Pickling - Removing surface oxides from metals by chemical or electrochemical reaction.

Pill Press - A press capable of automatic repeat pressing, either mechanical or hydraulic.

Pill Pressing - Process used to shape carbide powder by compressing it between two punches in a die.

Pin Lok - A toolholder or insert which utilizes the hole in an insert and the pin in a toolholder for fixing the location and firmly holding the cutting tool insert in place in the toolholder. PIN LOK is an Adamas registered trademark.

Pin-Type Insert - An indexable cemented carbide cutting tool insert with a hole in the center used to locate the insert and clamp it in place by means of a pin in the toolholder. Need for a top clamp is thus avoided.

Pit - (Synonymous with void). Technically could be called macroporosity or large porosity. Holes .0016 inch (40 microns) or larger).

Pocket - See Insert pocket.

Popcorn Chip - Similar to corrugated; badly deformed but which scatter in all directions.

Porosity - Microporosity or very small voids, not interconnected. "A" porosity indicates diameters less than .0004 inch (10 microns) and "B" porosity indicates diameters of .0004 to .0016 inch (10 to 40 microns), "C" porosity is caused by excess carbon. The number of pores is compared to a standard and designated one (very few) to six. Thus, A-1 porosity is common in very good quality cemented carbide. Porosity is rated in accordance with ASTM B-276-54 (1972) entitled "Evaluating Cemented Carbides for Apparent Porosity".

Positive Geometry - Provides clearance from cutting edge; this tool is considered freer cutting.

Positive/Negative Geometry - This tool is a combination insert; the positive or negative geometry is determined by the molded top rake face of the insert. This tool is more economical and versatile due to this design.

Positive Rake - An insert (Code G in third position) that is ground on all surfaces to specified dimensions +/- 0.001 in. on cutting point and +/- 0.005. in. on thickness. (See Super-precision insert)

Pot - Socket to accept taper tools in a preset fixture.

Powder - A multiplicity of particles which make up a fine non-gritty material. In the carbide industry, powder is generally fine in nature and soft or non-gritty to the touch. In the powder metallurgy industry in general, however, the particles may be quite coarse so that powder could consist of granular, gritty particles which make a sand-like consistency of powder. Powder can refer to the individual metal binder powders, such as cobalt, nickel powder or the various carbide powders such as tungsten carbide powder or a mixture of the binder and the carbide powder into a material known as ready-to-press powder (see definition).

Powder Metallurgy - Making metal parts starting with metal powder which is at least compacted and sintered.

Power - The product of force times distance divided by time; it measures the performance of a given amount of work in a given time. It is expressed as foot-pounds per minute, inch-pounds per second, meter-kilograms per second, etc., or for electrical measurements in watts. One horse-power, 33,000 foot-pounds per minute, equal to 746 watts.

Precipitation Hardening - hardening caused by the precipitation of a constituent from a supersaturated solid solution.

Precision Insert - An insert which is ground on surfaces to specified dimensions +/- 0.001" on cutting point and +/- 0.005" on thickness. (See Super Prevision Insert)

Preform - To shape or form in the green stage, generally presintered.

Prehoned - Machine-honed to a uniform size by the insert manufacturer. (See Honing)

Preset - Method of setting a tool length,depth or diameter, off of the machine.

Preset Fixture - Device used to preset tools off of the machine.

Presinter (Dewax) - The low temperature firing of pressed, extruded or otherwise consolidated powder, in order to achieve an increased green strength over the pressed state to remove the lubricant temporarily added to hold the powder together, and to get the part enough strength so that it may be shaped without resorting to grinding in a green condition. (See definition of preform.)

Pressed - Compacted, and formed under pressure. Usually requires a die to hold the powered material and a press to compact through applied pressure.

Press Fit - To force two parts together so that the friction holds the part readily in place. A cylindrical die is often press fitted into a steel holder. In a press fit, an inserted part can be removed under moderate force contrasted to a wringing or shrink fit.

Pressing - Forming a powder metal part with expressive force. (See compacting)

Production Shop - Machine shop producing multiple runs of the same parts.

Profiling - Machine operation where the tool does not move parallel to the workpiece but follows contours.

Pull-Out - The removal of chunks of grains of material from cemented carbide due to heat, oxidation or corrosion attack. This may occur in cutting tools and wear parts. It can occur due to defective quality cemented carbide, but is more often due to the fact that the carbide grade is not adequately heat resistant for the application.

Punch - Compacting tool used in combination with a mold.

Push-On-Arbor - A device used to hold in the I.D. of a part for inspecting purposes.

Qualified Holder - Tool holder manufactured to close tolerances for use on N/C machines, usually +/- .003.

Quick Change - A toolholding system for quick change over of taper tooling.

R.P.M. - Revolutions per minute. (See also Speed)

Radial Rake - The angle between the tooth and a rake line passing through the cutting edge in a plane perpendicular to the cutter axis.

Rake - The angle of inclination between the face of the cutting tool and the work. If the face of the tool lies in a plane through the axis of the work (n a round workpiece), the tool is said to have zero, or neutral, rake. If the inclination of the tool face makes the cutting edge keener or more acute than when the rake angle is zero, the rake is defined positive. If the inclination of the tool face makes the cutting edge less keen or more blunt than when the rake angle is zero, the rake is defined s negative. (See Back Rake, Side Rake)

Rake Angle - (Back angle) - slope of the back of the insert in relation to the workpiece centerline.

Rake Face - See Face.

Rapid Change - A tap cartridge system for quick change over of tap sizes.

Rigid Clamp - Air operated spindle brake.

Ready to Press Powder - Powder formulated into the condition where it is ready to be compacted either in automatic presses, large mold hydraulic presses, hydrostatic presses or extrusion, whatever the case may be. Granulated powder, pelletized powder, hydraulic powder and lubricated powder are all forms of ready to press powder.

Reclaim - Reclaim is powder removed from dust collectors utilized in preforming operations, from broken pieces in preforming, or from broken pieces in pressing. It consists of clean broken pieces which have not been sintered or clean loose powder. Reclaim, therefore, is material generated internally by the carbide manufacturer and which may be used in tire studs or utility grades or after suitable special reprocessing in regular standard grades.

Recrystallization - (1) The change from one crystal structure to another, as occurs on heating or cooling through a critical temperature; (2) The formation of a new, strain-free grain structure from that existing in cold worked metal, usually accomplished by heating.

Reduction of Area - (1) Commonly the difference, expressed as a percentage of original area, between the original cross-sectional area of a tensile test specimen and the minimum cross-sectional area measured after complete separation; (2) The difference, expressed as a percentage of original area, between original cross-sectional area and that after straining the specimen.

Relief - The clearance angle behind or below the cutting edge, allowing the cutting edge to be forced into the work. It is sometimes divided into primary relief (adjacent to the cutting edge) and secondary relief (beyond the primary relief). Depends on both the clearance angle of the insert and the insert attitude in the toolholder.

Right Hand Holder - Tool holder which, when viewed from the front, has an insert on the right hand side.

RMS - (Root Means Squared) - Old method of determining surface finish; some measuring devices using RMS still in use.

Rockwell - System for measuring material hardness.

Rockwell Hardness Test - A test for determining the hardness of a material based upon the depth of penetration of a specified penetrator into the specimen under certain arbitrarily fixed conditions of test.

Roller Drive Floating Holder - A tool for holding reamers, allowing radial float only. Can be used in rotating applications.

Rotating Tool - Cutting tool which revolves during cutting.

Rough Cut - Initial passes of cutting tool characterized by heavy metal removal rates; high feed and depth of cut.

S.F.M. - Surface feet per minute. (See also S.F.P.M.)

S.F.P.M. - Surface feet per minute. (See also S.F.M., Speed)

Safe Drive - A series of clutch discs in a tap chuck that allows the chuck to spin when a reset torque is reached.

Sandwich Braze - A method of brazing with a copper shim between two layers of brazing material. In brazing, the copper shim stretches relieving the stress caused by the differential or expansion between the carbide blank and the steel tool shank. This method is often used to prevent cracking large pieces of carbide due to brazing stress.

Scott Density - The apparent or bulk density measured by the use of a Scott Volumeter now formed in accordance with ASTM Standard B329-70 to measure of the weight of powder occupying one cubic inch in the loose unpacked condition. (See also Apparent Density)

Screw Stock - Free machining bar, rod or wire.

Seam - (1) On the surface of metal, an unwelded fold or lap which appears on a crack, usually resulting from a defect obtained in casting or in working; (2) Mechanical or welded joints.

Season Cracking - Cracking resulting from the combined effects of corrosion and internal stress. A term usually applied to stress-corrosion cracking of brass.

Seat (Anvil, Shim) - A removable part of a toolholder, designed to support the cutting insert. (See Anvil, Insert Package)

Self-Releasing Tap Chuck - A tap chuck that disengages at the same place, ideal for holding pipe tap depth.

Sfm (Surface feet per minute) - Peripheral velocity of the workpiece at cutting radius. (See Cutting Speed)

Shank - The projecting portion of a milling cutter which locates and drives the cutter from the machine spindle adapter. Also, the main body of a single point tool or toolholder.

Shear Angle - The angle between the shear plane and the face of the work (in milling), or a tangent to the work at a point on the work surface directly above the chip.

Shear Plane - Metal is separated in the machining operation by a shearing action which takes place in the plane passing through the cutting edge of the tool at the surface of the work directly above the chip. The plane is called the shear plane.

Shell End Mill - An end mill without an integral shank which is designed to mount on the machine spindle by means of a stub arbor or adapter and has a mounting hold and keyway across the back for this purpose. (See also End Mill)

Shim - Carbide pad used under indexable insert that provides flat seating area for insert and adds strength to overall tool performance.

Shim Screw - This screw (usually a flat head screw) locks shim in place to prevent movement during indexing.

Shrink Fit - The method of holding together two parts which is frequently done when a carbide insert is placed into a steel holder where high forces are encountered and a press fit is inadequate. In the shrink fit, the steel generally heated and placed over the carbide insert such that on cooling, the steel shrinks tighter and compresses the carbide insert to hold it very tightly in place.

Shrinkage - The reduction in dimensions or size that the piece undergoes from the pressed or presintered condition to the final sintered condition. This change in size is generally on the order of 0.2 inches per inch. Thus, a piece pressed 1.2 inches long will be 1.000 inches long approximately when finally sintered. Shrinkage occurs in all three dimensions, but may vary slightly in the amount of shrinkage in length to shrinkage in thickness.

Side- Cutting -Edge angle (Lead angle, Attack Angle) - The angle between the side cutting edge and the projected side of the shank or holder. (See Lead Angle)

Side Mill - A peripheral milling cutter having a keyway in the mounting hole and designed to cut on the periphery and one or both sides. A full side mill cuts on both sides. A half side mill cuts on one side only and must be designated right or left-handed. (See also Full Side Mill, Half Side Mill, Peripheral Mill).

Side Rake - The inclination of the face of the tool away from the side cutting edge. It is measured in a plane perpendicular to the top plane of the tool and the side cutting edge. (See Rake)

Side Relief Angle - The angle between the side flank immediately below the side cutting edge and a line drawn through the side cutting edge perpendicular to the base.

Single Point Tool - A cutting tool having only one cutting edge. A lathe tool.

Sinter (Hydrogen) (Vacuum) - Sintering is the operation for heating a powder compact so that it shrinks and densifies to a density approaching its void free or fully dense condition. The sintering results in a consolidation for the powder particles into a coherent or conglomerate mass. In the case of carbide, some liquid forms around the periphery of the particles during the sintering operation, and this is referred to as liquid phase sintering. The term, hydrogen sintering, refers to the fact that it is done in a hydrogen atmosphere. Vacuum sintering is carried out in a vacuum.

Slit - A macroscopic to sub-macroscopic void which may be detected in a fractured piece of carbide with the naked eye or with up to 10x magnification and characterized by a saucer-like or cigar-like shape. A slit is a void which has been caused by incorrect pressing or by some condition in the powder which resulted in the void being present in the pressed condition.

Sleeve - Bushing similar to a collet except double angles are inside for I.D. gripping of parts.

Slotting Cutter - A full side mill designed to cut slots. (See also Full Side Mill, Staggered Tooth Cutter)

Soft Grade - Grade containing high percentage of binder material.

Solution Heat Treatment - Heating an alloy to a suitable temperature, holding at that temperature long enough to allow one or more constituents to enter into solid solution, and then cooling rapidly enough to hold the constituents in solution. The alloy is left in a supersaturated, unstable state and may subsequently exhibit quench aging.

Special Item - An item manufactured for a particular customer according to his specifications.

Specific Gravity - The ratio of a material's weight to the weight of an equal volume of water.

Speed (Cutting Speed, Surface Speed) - See Cutting speed.

Speed Indexer - Air operated positioning device available in 400, 450, and 600.

Spindle Speed - Speed of machine expressed in RPM.

Spring Clip - Ring that hold the nose ring inside the nosepiece.

Square Shoulder - 90° angle machined into a workpiece.

Stabilizing Treatment - Any treatment intended to stabilize the structure of an alloy or the dimensions of a part. (1) Heating austenitic stainless steels that contain titanium, columbium or tantalum to a suitable temperature below that of a full anneal in order to inactivate the maximum amount of carbon by precipitation as a carbide of titanium, columbium or tantalum; (2) Transforming retained austenite in parts made from tool steel; (3) Precipitating a constituent from a nonferrous solid solution to improve the workability, to decrease the tendency of certain alloys to age harden at room temperature or to obtain dimensional stability.

Staggered Tooth Cutter - A full side mill with the side cutting edges alternating from side to side. (See also Full Side Mill, Slotting Cutter)

Standard Item - An item which is manufactured and kept in stock for customers.

Standardization - A program designed to maximize the versatility and productivity of shop tooling while minimizing the number of tools used.

Starting Point - Feed, speed and depth of cut used during trial cut; depending upon the results, operating conditions are either increased or decreased.

Steel Cutting Grade - Metal cutting grade of tungsten carbide; binder is usually titanium or tantalum carbide.

Stock (Grind Stock) - The additional material added to dimension to allow it to be ground to size.

Strength - The stress or force per unit area which material can withstand under tension, compression and bending, etc.

Stress Cracks - Cracks caused by a stress too high for the material.

Stress Relieving - Heating to a suitable temperature, holding long enough to reduce residual stresses and then cooling slowly enough to minimize the development of new residual stresses.

Stub Arbor - A device to mount in or on the spindle of a machine tool, and which is designed to carry and drive a shell end mill or face mill. (See also Arbor)

Superprecision Insert - An insert ground on all surfaces to a closer tolerance than a precision insert. There are several classes of superprecision inserts, with tolerances on the nominal point dimension ranging from +/-0.001 in. to +/-0.005 in.

Sup-R-Dex - An adapter to extend a tap further out from the chuck face.

Surface Finish - Physical characteristics of the machined workpiece surface.

Tangential Tool - A small tool or bit held in holders so that the bit is nearly radial or at a tangent to the worked surface. The tool is provided with a chipbreaker.

Tantalum Carbide (TaC) - A compound of equal parts by atomic weight of tantalum and carbon. The carbon content of tantalum carbide is 6.22 weight percent, and the tantalum content 95.78 weight percent. Tantalum carbide has a melting point of 3880°C (7000°F) and density of 14.48 g/cm^3. The Vickers hardness is 179_ kg/mm^2. Tantalum carbide consists generally of a fine powder composed of hard particles generally with an average particle size in the range of 1 to 5 microns. It is used as an additive to cobalt cemented tungsten carbide in steel cutting grades for metal machining applications and in wear parts to enhance hot hardness.

Tension - The ability of a tap chuck to compensate axially when tap lead and machine feed are not the same.

Tensile Strength - In tensile testing, the ratio of maximum load to original load to original cross-sectional area. Also called ultimate strength.

Tenthset - A system of adjusting boring tools within .0001/inch increments for single point cutting.

Thermal Cracks - Separations in the cutting tool generally visible in the crater or top face of the cutting tool due to high temperatures encountered in some metal cutting operations. To decrease thermal cracking effects, a more heat resistant grade is selected.

Thermal Deformation - Insert distortion caused by excessive heat.

Thermal Expansion - The increase in size caused by heating.

Throwaway Insert - A carbide shape ground on top and bottom or ground all over which can be mechanically held in a toolholder and indexed or inverted for new cutting surfaces. It is discarded after all cutting edges are used.

Tightening Fixture - Pot attached to a bench or held in a vise to aid in tightening or loosening collet chucks, shell mill holders, etc.

Tip - A blank when brazed on a shank to form a cutting tool or wear part. Often used synonomously with insert.

Titan 60 - A patented cemented titanium carbide formulation used primarily for cutting tools employing moderate to high speeds for semi-rough to semi-finish machining primarily of steels. TITAN 60 is an Adamas registered trademark.

Titan 80 - A patented cemented titanium carbide formulation used for semi-finish and finish machining at moderate to high speeds primarily for steel. TITAN 80 is an Adamas registered trademark.

Titanium Carbide (TiC) - An intermetallic compound of titanium and carbon consisting of equal parts by atomic weight of titanium and carbon. The compound consists of 20.05 weight percent carbon and 79.95 weight percent titanium. It has a melting point of 3,140°C (5,700°F), a Vickers harness of 3200 kg/mm^2, and a density of 4.94 g/cm^3. Titanium carbide powder, generally in the range of 1 to 5 microns in size, is used for making "titanium carbide" materials. The latter actually refers to cemented titanium carbide which consists of titanium carbide crystals imbedded in a matrix of nickel with molybdenum carbide frequently added. Thus, the titanium carbide tools would refer to a cemented titanium carbide as contrasted to the titanium carbide compound which is generally found in powder form. Titanium carbide is also a constituent added to tungsten carbide cobalt for steel cutting grades of cemented carbide cutting tools. Titanium carbide is sometimes used as a thin coating (0.0002" approximately) on cemented tungsten carbide to improve metal cutting performance of throwaway inserts.

Tolerance - The total amount of variation permitted from a specified dimension. It may be expressed as a plus, minus or both.

Toolholder - A tool component that mechanically holds the insert and that, in turn, is mechanically affixed to a tool-carrying component of the machine tool, such as a turret. Loosely, a tool-shank for insert-type lathe tooling.

Tool Overhang - Distance tool holder projects beyond tool part or other holding mechanism.

Tool Shank Socket - Pot to accept taper tools in a preset fixture.

Tool Steel Cutting Tool - Drill or turning tool whose composition is entirely high carbon alloyed steel.

Top Clamp Holder - Tool steel shank which holds the insert in place with a top clamp.

Top Rake (Back Rake) - The angle of inclination of the tool face away from the end cutting edge. It is measured in a plane perpendicular to the base of the tool and parallel to the side cutting edge. (See Rake)

To Size - The nominal dimensions made to stand manufacturing tolerance without grind stock allowance.

Toughness - The ability to withstand breakage impact.

Tough Pitch Copper - Copper containing from 0.02 to 0.05% O, obtained by refining copper in a reverberatory furnace.

Transverse Rupture Strength - Breaking strength of a material in a standard bending test (ASTM B406-70 for carbides). Measured in pounds per square inch (psi) in the US. Generally used as an indication of the toughness of a cutting -tool material although certain limitation exist. For example, the test is static and may not provide a good prediction of performance in a dynamic cutting mode.

Troubleshooting - Systematic approach to problem solving.

Tumble - Abrasively finished by rotating in a drum or vibratory means in a drum.

Tungsten Carbide (WC) - Tungsten carbide is an intermetallic compound consisting of equal parts by atomic weight of tungsten and carbon. Tungsten carbide consists of 6.13 weight percent carbon and 93.87 weight percent tungsten. It has a density of 15.77 g/cm^3 and decomposes at 2,750^{o}C (4,970^{o}F). Tungsten carbide particles have a very high hardness with a Vickers hardness of 2800 kg/mm^2. The tungsten carbide powder that is used in making cemented tungsten carbide is generally composed of tungsten carbide particles ranging from a minimum size of less than one micron up to two microns. Sometimes tungsten carbide is used in reference to the cemented tungsten carbide material with cobalt added and/or with titanium carbide or tantalum carbide added. Thus, tungsten carbide may be used to refer to pure tungsten carbide as well as co-bonded tungsten carbide which may or may not contain added titanium carbide and/or tantalum carbide.

Tungsten Carbide (W_2C) - An intermetallic compound of tungsten and carbon in which there are two parts by atomic weight of tungsten to one part by atomic weight of carbon. The carbon content is 3.16 weight percent and the tungsten content 96.84 weight percent. This form of tungsten carbide has a melting temperature of 2,800^{o}C (5,070^{o}F) and a density of 17.20 g/cm^3. It has a Vickers hardness of 2900 kg/mm^2. It is also used in flame plasma sprayed hard facing compositions. The compound W_2C is undesirable in making regular cemented tungsten carbide and results in carbon deficiency.

Tungsten Free - A cemented carbide formulation containing no tungsten or tungsten carbide. Cemented titanium carbide formulations are often tungsten free.

Turning - A machining process for making external surfaces of revolution by the action of a cutting tool or rotating workpiece. This is usually done on a lathe.

Turning Machine - A machine which performs the operation of turning; a lathe.

Turret - A pivoted, removable device or machine tool which contains a number of tool holders.

Ultrasonic Inspection - A non-destructive "pulse-echo" test method for locating the position and extent of internal defects in metals by the scalar measurement of incident (pulse) and reflected (echo) sound waves.

Undersinter - A condition in a piece of carbide where the sintering temperature and time has been inadequate to bring the piece to full and proper density and where sintered carbide is characterized by lower than acceptable table density and higher than acceptable porosity.

Utility Insert - An insert ground on the top and bottom surface only, with a common tolerance of +/-0.005 in.

Wear - Displacement of carbide particles caused by abrasion or adhesion and diffusion.

Wear Land - A flat area worn on the relieved flank face of the insert, below or behind the cutting edge. The depth of wear affects size and finish, and the width of the wear land is a good indicator for comparing insert performance or for determining the proper time to change or index the tool. (See Abrasive Wear, Flank Wear)

Welding - (1) Joining two or more pieces of material by applying heat, pressure or both, with or without filler material, to produce a localized union through fusion or recrystallization across the interface. The thickness of the filler material is much greater than the capillary dimensions encountered in brazing; (2) May also be extended to include brazing.

Welding Stress - Residual stress caused by localized heating and cooling during welding.

Working Angle - Those angles between tool and work that depend not only on the shape of the insert but also on its position with respect to the work as determined by toolholder.

Yield Point - The first stress in a material, usually less than the maximum attainable stress, at which an increase in strain occurs without an increase in stress. Only certain metals exhibit a yield point. If there is a decrease in stress after yielding, a distinction may be made between upper and lower yield points.

Yield Strength - The stress at which a material exhibits a specified deviation from proportionality of stress to strain. An offset of 0.2% is used for many metals.

Zero Rake (Neutral Rake) - See Rake.

A Special Thanks to: **Victor Yessayian - Technical Sales Representative Kennametal**
Ed DeYoung - Metalworking Systems Engineer Kennametal

For their help with the GLOSSARY section. It is very much appreciated........

Kennametal users, please memo your corrections, changes, and additions to RB6NREL.

finding better ways

KENNAMETAL®

North American Service Centers

California
6015 East Randolph Street
Los Angeles, CA 90040-3486
Phone 800-446-7738
Fax 800-732-5760

Connecticut
1095 Day Hill Road
Suite 150
P.O. Box 800
Windsor, CT 06095
Phone 800-446-7738
Fax 800-847-0004

Illinois
2150 Western Court
Suite 300
Lisle, IL 60532
Phone 800-446-7738
Fax 800-962-4910

Michigan
31572 Industrial Road
Suite 300
P.O. Box CN-5100
Livonia, MI 48151-0510
Phone 800-446-7738
Fax 800-321-7290

North Carolina
4301 Westinghouse Boulevard
Charlotte, NC 28273
Phone 800-446-7738
Fax 704-588-9687

Ohio
Duke World Park, Building #6
5577 Spellmire Drive
Cincinnati, OH 45246
Phone 800-446-7738
Fax 800-932-2399

Mexico (Sales Offices)
Kennametal de Mexico S.A. de C.V.
Boulevard de la Santa Cruz No. 13
Fracc. Boulevares, C.P. 53140
Naucalpan
Estado de Mexico
Phone (52) 5 572 96 68
Fax (52) 5 393 33 86

Kennametal de Mexico
Morones Prieto 2805 Pte. 6-3
Col. Loma Larga 64710
Monterrey, N.L., Mexico
Tel: 011-52-8-319-0244 al 47
Fax: 011-52-8-319-0248

Canada (Kennametal Ltd.)

Alberta
Unit No. 2, 9750-51 Avenue
Edmonton, AB T6E 0A5
Phone 800-446-7738
Fax 403-436-9364

British Columbia
Unit 11, 3511 Viking Way
Richmond, BC V6V 1W1
Phone 800-446-7738
Fax 604-273-8380

Manitoba
1226 Sherwin Road
Winnipeg, MB R3H 0V3
Phone 800-446-7738
Fax 204-633-9162

Ontario
115B Matheson Boulevard West
Suite #211
Mississauga, ON L5R 3L1
Phone 800-446-7738
Fax 905-568-2955

Quebec
2244 - 46th Avenue
Lachine, PQ H8T 2P3
Phone 800-446-7738
Fax 514-636-1756

Kennametal Inc.
Metalworking Systems Division
P.O. Box 231
Latrobe, PA 15650

Customer Application Support Phone 800-835-3668

International Locations

Argentina
Australia
Austria
Belgium/Luxembourg
Brazil
Canada
Chile
China
Colombia
Czech Republic
Denmark
Dominican Republic
Egypt
Finland
France
Germany
Greece
Hong Kong
Hungary
India
Israel
Italy
Japan
Korea
Malaysia
Mexico
Netherlands
New Zealand
Norway
Pakistan
Peru
Philippines
Poland
Portugal
Puerto Rico
Russia
Singapore
South Africa
Spain
Sweden
Switzerland
Taiwan
Thailand
Turkey
United Kingdom
Venezuela

Customer Service (United States and Canada) Phone 800-446-7738

A96-59 (5) J7

printed in U.S.A.