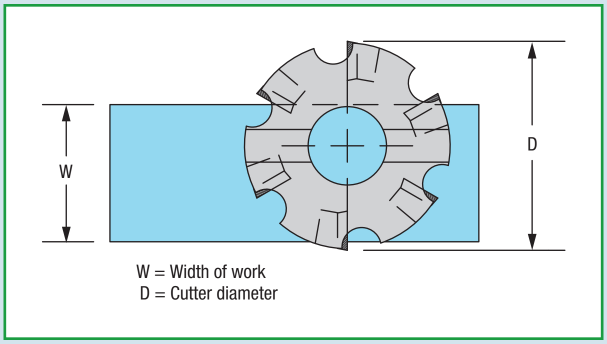
Selection of Correct Cutter Diameter

Select a cutter diameter greater than the workpiece width by a ratio of approximately 1.5 to 1. This will ensure that each insert enters the cut without the frictional, no-chip phase which occurs when attempting to cut the full cutter diameter. Also, the insert leaves the part without reducing the chip down to zero. These benefits can greatly extend the insert life.

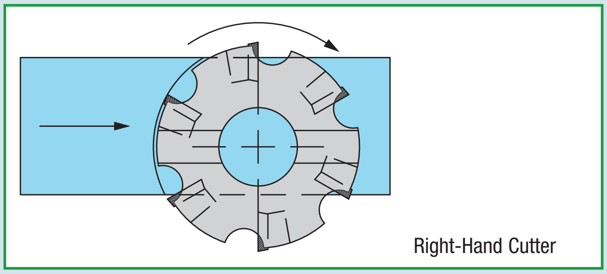
With smaller, low horsepower machines it will be better to select a smaller cutter and take two passes rather than a large diameter cutter forced to operate at low tooth loads (feed rates) to avoid stalling of the spindle.



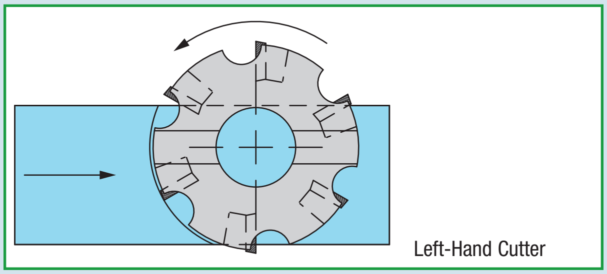
Choose a cutter diameter approximately 1.5 times the workpiece width.

Hand of Cutters

A *right-hand cutter* is one which, when viewed from above, rotates clockwise relative to the workpiece.



A *left-hand cutter* is one which, when viewed from above, rotates counterclockwise relative to the workpiece.



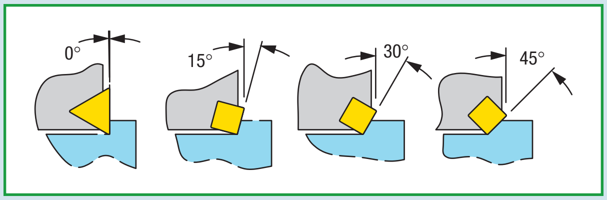
Lead Angle

The lead angle of a milling cutter is not intended for producing a specific angle on the work. In fact, because of compound angles, a given lead angle will not produce that angle exactly.

The purpose of lead angle is to thin the chip while absorbing a given depth of cut over a greater portion of the insert edge. This results in improved tool life and, for a given horsepower, a greater depth potential.

For example, 30° lead angle is a good choice for face milling in general purpose applications.

The exception to the previous statement is the 0° lead cutter, sometimes called a 90° cutter, which is designed for milling to square shoulders and producing a 90° corner.

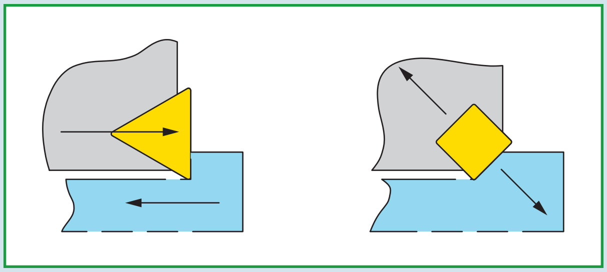


Lead Angles and Cutting Forces

The lead angle of a milling cutter has a direct effect upon the cutting forces being presented to the workpiece, cutting tool, and machine.

The resultant force is always directly perpendicular to the cutting edge. A lead angle may, therefore, be a major consideration in how we want to direct the forces.

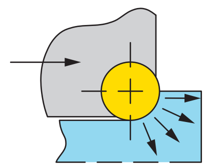
For example, in a thin section workpiece, a high lead angle may cause deflection since there is more tendency to “push” the part away from the cutter. On the other hand, a 0° lead cutter has more deflective force on the machine spindle.



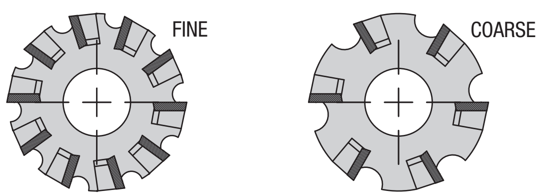
Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

The Round Insert Cutter

The exception to the rule in lead angle cutting forces is the round insert. With a round insert, the lead angle is entirely dependent upon the depth of cut. As the depth increases, the lead angle decreases. If cutting half the diameter deep, there is effectively 0° lead angle.



In the milling of work hardening materials such as Inconel, and using a round insert cutter, there will be a direct relationship between depth of cut and speed of development of notch wear. The shallower the cut, the slower the notch wear.



Pitch

The pitch of a milling cutter refers to the numbers of inserts placed into a given diameter.

Cutters for cast iron are often closer pitch to allow the maximum number of teeth to be engaged at one time for smoother cutting, and because cast iron does not need large gullet for the discontinuous chips produced.

For general use, choose a fairly coarse pitch. A guide would be diameter plus 2, i.e., a 6" cutter with 8 inserts, etc.

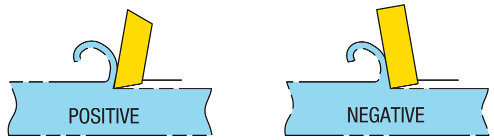
Negative Versus Positive Geometry

In an indexable cutter, the negative insert is the only one which permits the insert to be turned over and used on both sides. It is the most economical style. Also, it is the strongest insert because all edges are 90° to the faces.

On the minus side, the negative rake tool produces higher cutting forces when compared to the positive rake.

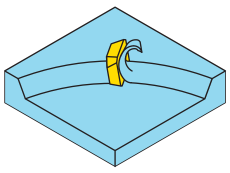
In general, use negative rakes for cast iron, interrupted cuts, and on rigid high horsepower machining for steels.

Use positive rakes for aluminum, titanium, copper, most stainless steels, thin or easily deflected parts, steels, and nickel alloys.



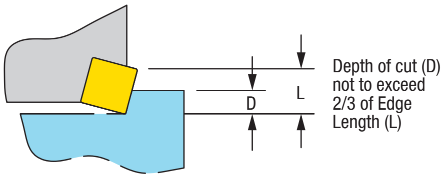
There are many milling cutters with a combination of positive and negative rakes often called shear-angle design. These cutters offer some of both worlds, although inserts are essentially like positive inserts and cannot be turned over. Shear angle cutters do provide continuous chip ejection since the axial rake behaves much like a helix in a flute and takes the chip up and away from the finished surface.

These cutters work well in heavy duty operations with wide widths of cut—especially if combined with a 30° lead angle.



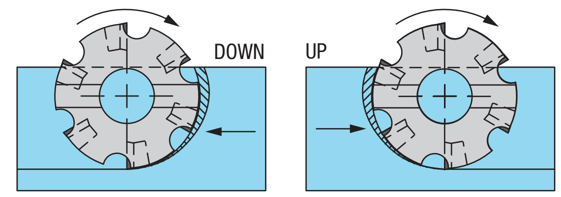
Depth of Cut

It is a good general rule not to allow depth of cut to exceed 2/3 of the cutting edge length. Remember that in lead angle cutters the cutting edge length in use is not the same as the depth of cut.



Up Milling and Down Milling

This refers to direction of rotation relative to the feed.



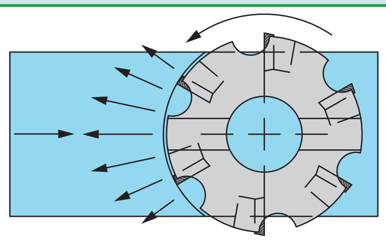
Down (climb) Up (conventional)

With a modern machine in good condition, down milling will give the best results. This is because the thickest section of the chip is against the insert to avoid welding, and pressure is progressively relieved towards the finished surface.

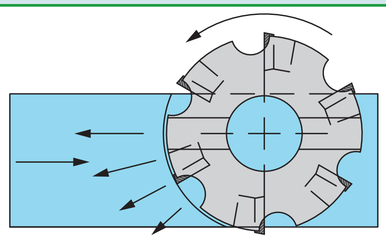
In up milling, friction and pressure build up before the chip starts to form, causing premature edge wear. It should be in rare cases that up milling is needed. This could be, for example, on an older machine with backlash in the table feed.

Cutter Positioning

Central positioning of the cutter can give rise to vibration if any spindle play is present. This is because of an alternating radial force pushing against the spindle.



Placing the cutter off center will always be a better situation to avoid chatter and vibration and also to improve tool life.

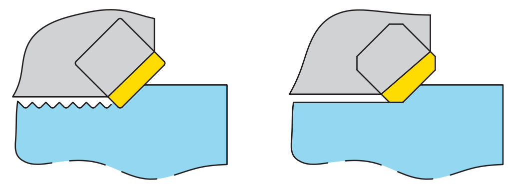


When moving off center, the path of cut is longer since each insert now sweeps a longer arc with each revolution. This may have a measureable impact on tool life, and cutting temperature will tend to increase.

Seek a happy medium by moving off center in small increments until vibration is controlled.

Surface Finish

In a milling cutter the finish is produced by the highest insert. Since variations exist in the body and the inserts, it is inevitable that some inserts will be higher than others. If the inserts have small corner radii, for example, the highest insert will cut the track and this will determine the finish.

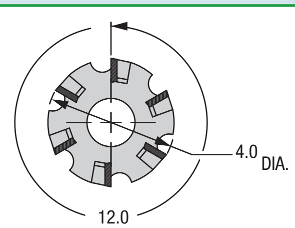


For this reason, most inserts designed especially for milling, use flats on the insert rather than a radius. In this way, the highest insert produces a wiping effect removing the variances of the other inserts and leaving a much improved finish. “Wiper” inserts installed in a few stations can be used for this purpose as well as “finishing” inserts which are available for certain cutters in the Greenleaf line.

The 4" Reference for Speed Calculations

Recommended cutting speeds are usually given in surface feet per minute (SFM). Sometimes a problem exists in converting SFM to the correct RPM (revolutions per minute) for a given cutter diameter.

A very easy way to make a quick approximate calculation is to use a 4" cutter as a base of reference. Since a 4" cutter has a circumference of approximately 12" or 1 foot $\frac{(\pi \times D'')}{12}$ = Cft the correct RPM for a 4" cutter is the same as required speed in SFM, i.e,100 RPM = 100 SFM.



This makes it easy to make a mental calculation for most popular cutter diameters

Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

For Example:

An 8" cutter has 2x the circumference. Therefore, 100 RPM=200 SFM. A 2" cutter has half the circumference. Therefore, 100 RPM=50 SFM and so forth.

If you want to make an accurate calculation, the formula is: $\text{SFM} = \dfrac{\pi \times d \times \text{RPM}}{12}$

Speed rate recommendations are based upon the material to be machined and the cutting tool material which will be used, i.e., carbide, coated carbide, ceramic, silicon nitride, etc.

Feed Rate Calculation

One problem encountered in milling cutter feed rate considerations is that while most milling cutter manufacturers make recommendations in load per tooth or feed per tooth, the machine is calibrated in *inches per minute*. It is, therefore, necessary to do a little simple math to get the answers required.

In turning, these problems do not exist since only one insert is involved, and the machine is calibrated in feed per revolution. Feed per revolution is the same as feed per tooth when there is only one insert, so we simply plug in the recommended feed.

With a milling cutter, the feed per tooth is controlled by three factors. These are:

1. The feed rate or table advance in inches per minute.
2. The spindle speed in revolutions per minute.
3. The number of inserts in the milling cutter.

We must make a calculation in order to find out the really critical information needed, i.e., what is the feed per tooth or how much work are we asking each insert to perform? Too little work is more often a problem than too much.

If the feed per tooth is very small, let us say less than .003", then abrasive wear is accelerated. No real chip is produced to take away the heat.

On the other hand, if high feed rates are used and the cutter has many teeth, then horsepower available may be insufficient. This is an important consideration in selecting a cutter, especially larger diameter cutters with fine pitch. Here are the equations you will need to make your calculations:

T = Number of teeth
FPT = Feed per tooth
IPM = Inches per minute
RPM = Revolutions per minute
π = 3.1416

$$\text{Feed per tooth} = \frac{\text{IPM}}{\text{T} \times \text{RPM}}$$

$$\text{Feed per revolution} = \frac{\text{IPM}}{\text{RPM}}$$

$$\text{Inches per minute} = \text{FPT} \times \text{T} \times \text{RPM}$$

$$\text{Revolutions per minute} = \frac{12 \times \text{SFM}}{\pi \times d}$$

These calculations can also be readily made using the Greenleaf milling calculator available free of charge upon request from your local representative or directly from Greenleaf Corporation (800-458-1850). This calculator also displays horsepower needed at the spindle for a given cut. This takes into account width and depth as well as speed and feed for a given cutter together with the machinability of the material to be machined, often referred to as the "K" factor.

It is a good starting point to know that a mild steel (150BHN) requires about 1 HP per cubic inch of material to be removed per minute.

The formula for cubic inches removed is:

Cu. ins. = D x W x IPM

Depth = .060

Width = 6 inches

IPM = 22 inches per minute

.060 x 6 x 22 = 7.92 cubic inches per minute
(or)
Approximately 8HP needed for steel 150 BHN

For any other material we can divide our answer by the "K" factor which is a machinability rating relative to 150BHN steel.

Greenleaf
Excelerating Solutions
800-458-1850

“K” Factors

Material	“K” Factor
Aluminum	4.00
Brass–soft	3.00
Brass–hard	2.00
Bronze–hard	1.40
Cast iron to 200 BHN	1.75
Cast iron to over 200 BHN	1.20
Malleable iron	1.50
Steel–100 BHN	1.40
Steel–150 BHN	1.00
Steel–200 BHN	0.85
Steel–250 BHN	0.83
Steel–300 BHN	0.80
Steel–400 BHN	0.65

HPc = Horsepower needed at spindle

D = Depth of cut

W = Width of cut

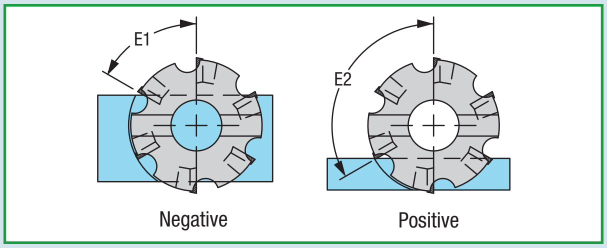
IPM = Inches per minute feed rate

K = K Factor

$$HPc = \frac{D \times W \times IPM}{K}$$

Angle of Entry

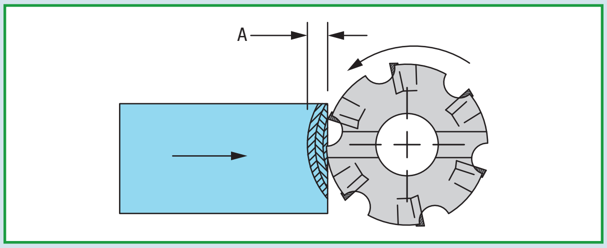
In face milling operations, the angle of entry can have a significant impact upon insert performance. A positive angle of entry can cause breakage or chipping, especially when using positive inserts. Positive angle of entry will occur when the path of cut is narrow relative to cutter diameter.



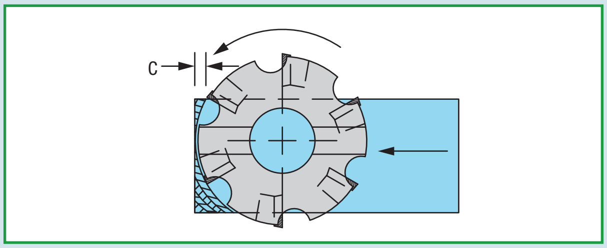
1. When the angle of entry (E1) is less than 90°, the initial impact occurs at a position behind the point of the tool. The insert has a greater section and is stronger here and better able to withstand the impacts.

2. When the angle of entry (E2) is greater than 90°, the initial impact between the insert and the part piece occurs at the point of the tool, which, especially in a positive rake milling cutter, is the weakest section of the insert. This can lead to insert failure.

Entering and Exiting the Cut



The angle of entry is always adverse as the cut commences. In the illustration, we can see that as the cutter travels through zone A, the angle of entry is changing. It starts out positive as the inserts first start to cut. As the cut progresses, it becomes less and less positive and eventually negative.



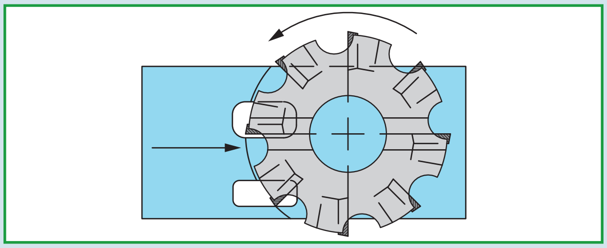
With a CNC machine, it is a worthwhile exercise to slow down the feed rate in zone A, especially with positive rake tools and hard to cut materials. As the cutter starts to break through at the end of the cut, another problem area is created in zone C. At this point, the cutter breaks through in the center, leaving two islands of material. Changes of entry angle occur which can result in insert problems. As in entry into the part, a reduction of feed rate can help alleviate chipping or breakage problems if they arise.

Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Interruptions

Milling is by definition an interrupted operation. In addition, as the cutter crosses voids in the part, changes of entry angle occur. This situation is usually too complex to define in absolute terms relative to a targeted solution. Recognizing this in interrupted parts, try to include some of the following features in the set-up to reduce impact:

1. Negative or negative/positive geometry
2. Use a lead-angle cutter (30° or 45°) if possible
3. Use an impact-resistant carbide grade
4. Use a cutter with medium or fine pitch
5. Keep the load per tooth on the low end



A Milling Cutter is a Series of Single-Point Tools

It is easy to lose sight of the fact that a milling cutter is nothing more than a series of single-point tools clamped into a rotating holder. If you always keep this in mind, you will be constantly reminded that what is most important to know is what is happening to each tool or insert.

The feed rate in inches per minute of machine table travel does not tell you anything important unless or until you calculate the feed per tooth. You cannot calculate the feed per tooth until you know the speed in revolutions per minute and how many teeth are in the cutter. Therefore, it should become second nature to ask, know, and consider the three "golden" variables:

1. How many inserts?
2. How many RPM?
3. What feed in inches per minute?

Use this formula to find feed per tooth:

$$\mathbf{FPT = \frac{IPM}{No.\ of\ Teeth \times RPM}}$$

Once you know the feed per tooth, as a very broad general guide, try to keep the feed above .003" per tooth and remember that horsepower limitations usually come into play long before most cutters reach the upper limit. Efficient metal removal will usually dictate working in the .005" to .010" per tooth range.

Some heavy-duty cutters can be used as high as .030" or more per tooth, but this will need a machine in the 50+ horsepower class – and a larger cutter could well use over 100 horsepower!

Greenleaf High Performance Mills* Set-up and Operational Procedures

1. Thoroughly clean all insert pockets.
2. Install the inserts, making sure that they are properly seated in the pocket, and torque the insert clamp screws to the correct torque as indicated on the body of the High Performance Milling Cutter.
3. Use Greenleaf High Performance Mills only on machines that have adequate shield guards.
4. Run the Greenleaf High Performance Mills using cutting parameters as recommended by Greenleaf Tech Team. Contact the Greenleaf Tech Team at: 814-763-2915 or by email: techteam@greenleafcorporation.com
5. For safety purposes, do not exceed the maximum RPM's etched on the High Performance Mill. Note: There are two max RPM numbers. One (the lower RPM number) is for using the mill with carbide inserts and the other is for usage with ceramic inserts.
6. Please refer to Greenleaf Engineering Bulletin – June 2, 2005: High Performance Mills / Maximum RPM's and Screw Torque for a complete listing of these specifications on Greenleaf High Performance Mills.

* *Includes Greenleaf milling products designated: C4, CP4, FMRN, FMRP, WSAN and the Excelerator™ Mills (WSRP, WSSP, WSRN, and WSTP)*