Lindsay Cutter Description System

L SN R - 10 - S N 6 - 30 - 6 (8)

Type of Line
M - Series
R - Series
LD - Series
LX - Series
LP - Series
CB - Series
LB - Series
L - Series

Direction of Cut
R - Right Hand
L - Left Hand

Insert Type
A - Parallelogram
C - 80 deg Diamond
L - Elongated
R - Round
S - Square
T - Triangle

Insert I.C.
In 1/8" Increments
- or -
Length of Cut
on L - Series' in
1/4" increments

Mounting Diameter
B - Bridgeport Shank (R8)
S - Screw On Shank or Bore
Diameter in 1/4" increments

Cutter Style
IS - Integral Shank (Pos or Neg)
SI - Integral Shank Step Cutter
I - Integral Shank (Neutral)
S - Slotter
SN - Double Negative Shell Mill
SP - Double Positive Shell Mill
SS - Stepped Shell Mill
IP - Integral Shank Endmill

Cutter Diameter In Inches

Insert Relief
B - 5-deg Positive
D - 7-deg Positive
E - 20-deg Hi-Positive
N - 0-deg Negative
P - 11-deg Positive

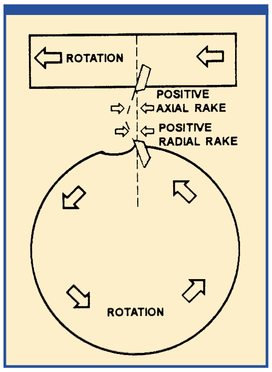
Lead Angle
Given In Degrees
- or -
Slot Width
(for Slotters Only)
Decimal Figure In
Parenthesis. i.e. (.5)

Number of Cutting Stations

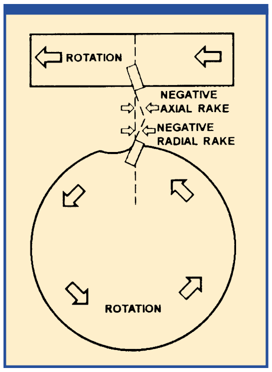
Milling operations performed with inserted tooth milling cutters are similar in many ways to that done by turning tools. The primary difference in milling with respect to the cutting edge is that the cut is always interrupted. As the cutter rotates and crosses across the part, each insert will enter and exit the cut at least one time per revolution. Not only does this cause a repeated impact situation on the cutting point, it also causes repeated heating and cooling of the insert itself lending it to be susceptible to thermal cracking.

Rake Angles

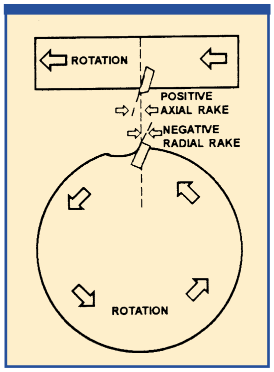
Rake angles for milling cutters are specified in two directions, axial and radial. Axial rake is the cutting insert's angle with respect to the central axis of the cutter/spindle assembly. Radial rake is the cutting insert's angle with respect to the periphery of the cutter. Common configurations include (a) Positive in both directions (b) Negative in both directions and (c) Positive in one direction and negative in the other.



Double Positive for milling cutters uses conventional positive rake insert geometries, this style of cutter is used where freer cutting action and lower cutting pressures are required. These cutters also produce a good work piece finish and often reduce workpiece deflection and vibration.



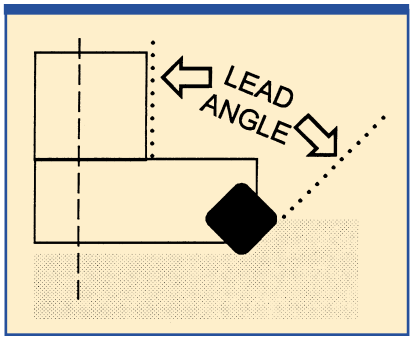
Double Negative rake cutters use conventional negative rake insert geometries. This style of cutter is used for maximum insert economy and in heavy-duty applications where adequate horsepower is available to handle the higher chip loads that are generated.



Negative-Positive rake angle cutters frequently use a negative rake insert to achieve negative radial geometry but have a positive relief angle ground under faceted corners to achieve a positive axial geometry. This style of cutter combines high insert strength with reduced cutting pressures. They are particularly well suited for steel milling as they tend to pull the chip up and away from the finished surface while simultaneously ejecting the chip radially. The 'Neg-Pos' cutter is a compromise design that provides the insert strength of the negative rake design with some of the cutting and chip flow characteristics of the positive rake design.

Lead Angles

Lead angles affect chip thickness and tool pressure as well as tool life. Lead angles are often dictated by the work piece geometry or the insert style required.



Where a square shoulder is necessary on the part, use a 0-degree lead angle. Triangle or parallelogram inserts must be used to provide for end clearance on the cutter. If the parts tolerance is somewhat open, a 1-degree lead may be used. It cuts close to the square shoulder and uses the more economical square insert.

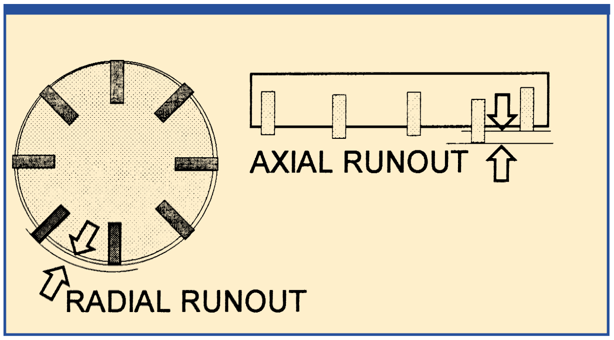
Increasing the lead angle reduces the cutters effective cutting depth. However, it does reduce cutting pressures and tool deflection. The shear angle is increased and the chip thickness reduced. Lead angles of 15 and 30 degrees are most commonly used, particularly on negative-positive style cutters. Lead angles of 45-degrees are effective for producing fine finishes with light depths of cut. Increasing the lead angle also reduces or minimizes the breakout of material or burr. Which may occur at the edge of the work piece being cut. On positive rake cutters, increased lead angle also tends to lift the chip away from the work piece thereby reducing the possibility of it scarring the freshly milled surface.

Cutter Runout

Runout is an important consideration in the application of any milling cutter. It is even more important when excellent surface finishes are required. There are two types of runout:

1. Radial – The difference in the locations of the insert cutting edges relative to the centerline of the cutter spindle.

2. Axial – The variation of cutting edge locations relative to the axial plane or the cutter face.



Fixed pocket milling cutters have no means of compensating for runout caused by manufacturing tolerances of inserts and cutter bodies. Therefore, there are several steps necessary to minimize the adverse effects of runout.

1. Use precision ground inserts.

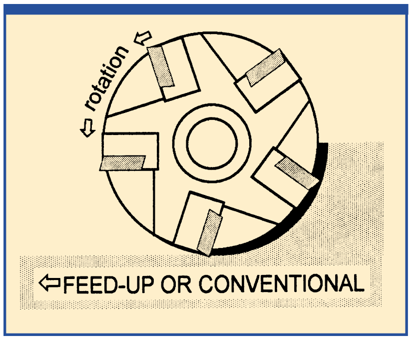
2. Proper cutter maintenance and setup.

3. Proper maintenance and setup

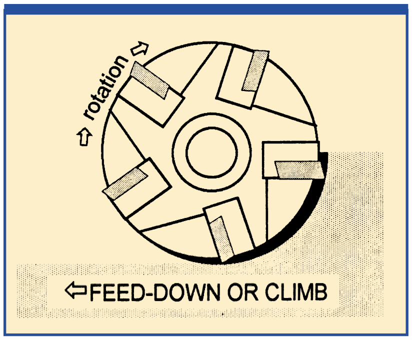
Once the cutter is in operation, reducing the federates relative to the spindle RPM may be the only way to minimize any problems attributed to runout.

"Up" versus "Down" Milling

Up or **conventional milling** is most often used on machines with no anti-backlash devices or on older conventional milling machines with a screw feed mechanism. In **up milling**, the cutter is milling *against* the direction of the feed. The insert enters from the previously milled surface and exits through the unmilled surface. This may be of advantage if the surface has heavy scale. On the other hand, **up milling** tends to lift the work piece during the cut. It can also result in work hardening at the milled surface on certain materials.

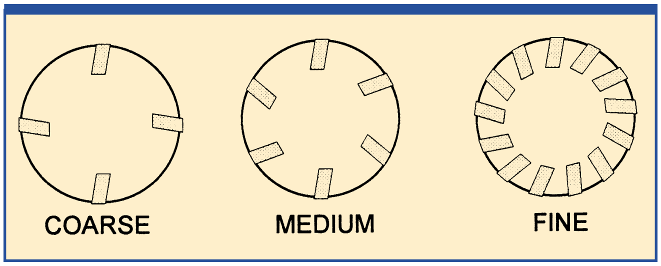


Down or **climb milling** is normally recommended for more modern ball screw fed machines or on machines that are equipped with a backlash limiting mechanism. In **down milling,** the cutter is milling in the *same direction* as the feed. The insert enters through the unmilled surface and produces a chip that becomes progressively thinner as the insert passes through the part. This action is often recommended when milling materials that tend to work-harden. **Down milling** also tends to place downward pressure on the work piece and forces the work in the direction of the feed.



Cutter Pitch

Cutter pitch refers to the number of cutting stations in a milling cutter. **Coarse**, **Medium**, or **Fine Pitch** are the normal classifications. Pitch is determined as the cutters are designed to allow for appropriate chip clearance, depths of cut, and feed rates per tooth.



Coarse pitch cutters are used for general purpose and heavy chip load milling operations. They have large chip gullets and adequate clearances to accept large depths of cut and high feed rates. **Coarse pitch** cutters generally draw less horsepower owing to fewer inserts engaged in cutting simultaneously.

Medium pitch cutters are for lighter feed rates and general-purpose work where it is desirable to have more than one insert cutting at a time. This cutter will reduce entry shock and cutting pressures while maintaining acceptable feed rates.

Fine pitch cutters have many inserts engaged in cutting at the same time. Chip load per tooth is reduced which in turn lessens the effects of deflection and chatter. Fine pitch cutters are effective in high production and on thin-walled parts, where edge breakout is a problem. The larger the number of teeth allows for high table feed rates in inches per minute while maintaining a light feed rate or chip load per tooth.

There are, however, two disadvantages to the **fine pitch** cutters. First, the increased number of inserts engaged in the cut at a given time will raise the horsepower requirements of the application. Secondly, the close spacing of the inserts reduces the clearance for the chip flow. The cutter is usable in light feed per tooth applications only.

Milling Formulas

TO FIND:	USE
Surface Footage per Minute (SFM)	.262 x Diameter x RPM
Revolutions per Minute (RPM)	SFM / (.262 x Diameter)
Feed per Tooth in Inches (IPT)	IPM (table feed) / # of teeth x RPM
Feed per Revolution (FPR)	IPM / RPM
Feed per Minute (FPM)	IPT x # of teeth x RPM
Metal Removal Rate (MRR)	IPM x width of cut (WOC) x depth of cut (DOC)

Calculating Power Requirements

The following formula can be used to approximate power requirements in milling:

$$\text{Horsepower @ cutter} = \frac{\text{IPM x WOC x DOC}}{\text{K Factor}}$$

The **K Factor** is the horsepower required to remove 1 in^3 per minute and can be estimated from the table at right…

VALID FOR CUTTERS WITH 0-DEGREE TOP RAKE

Material		Hardness Brinell HB	K factor
Non-alloy Carbon Steel	C<0.25%	125	1.14
	C<0.8%	150	1.05
	C<1.4%	250	0.96
Low-alloy Steel	Annealed	125-200	0.96
	Hardened	200-450	0.77
High-alloy Steel	Annealed	150-250	0.87
	Hardened	250-500	0.77
Stainless Steel	Ferr. Mart.	175-225	0.87
	Austenitic	150-200	0.77
Steel Castings	Non-alloy	225	1.23
	Low-alloy	150-250	1.09
	High-alloy	150-300	0.96
Very Hard Steel		>50 HRC	0.45
Malleable Cast Iron	Short chipping	110-145	1.41
	Long chipping	200-250	1.55
Grey Cast Iron	Low tensile	150-225	2.23
	Hign tensile	200-250	1.73
Nodular Cast iron	Ferritic	125-200	2.05
	Pearitic	200-300	1.41
Chilled Cast Iron		40-60 HRC	0.64
Aluminum alloy		100	3.73

Horsepower Info

Once horsepower at the cutter is determined, it is recommended to divide this result by .8 to compensate for the loss of power from the motor to milling machine spindle. Additionally, 2 other correction factors can be used with the above formula to further refine the horsepower requirements.

1) **Lead Angle**	0°	15°	45°
Correction Factor	1.0	.97	.71

2) **Axial Rake**	-10°	-5°	0°	5°	10°	15°	20°
Correction factor	1.30	1.07	0	.93	.87	.80	.74

Example: Milling high-alloy steel casting (.96 K factor from table)

4” WOC
125” DOC
22 IPM
45° lead face mill with 20° positive axial rake

Calculation: $\frac{22" \times 4" \times .125"}{.96} = 11.45$ HpC

Corrections: 11.45 x .71 x .74 = 6.02
6.02 / .8 = 7.5 machine horsepower

Getting Started

Unlike single point tools which work independently, each milling insert is dependent on the others in the cutter. Failure of one insert increases chip load on the following insert, which may result in severe breakage of all inserts.

To insure proper chip load for each insert, it is important that all inserts be set into the cutter properly and all be located to evenly distribute the cutting load.

When new inserts are placed in the cutter, or when an insert must be indexed due to wear, it is very important that the pocket and/or cartridge be thoroughly cleaned of any loose material which might prevent proper locating or seating of the inserts.

Milling inserts, like all indexable tooling inserts, must eventually be replaced due to failure by wear or manufacturing. The nature of this failure will show whether or not a satisfactory performance is being obtained.

Cutter Selection Guide

CUTTER STYLE	NEGATIVE RAKE						POSITIVE RAKE					HI-POSITIVE		
LEAD ANGLES	0	5	15	30	45	ROUND	0 OR 1	15	30	45	ROUND	1	5	30 OR 45
Free Machining Steels	1	1	2	2	2	2	1	2	2	3	2	-	-	-
400-500 Stainless	-	-	1	1	2	2	1	1	3	3	3	-	-	-
Gray Nodular Iron	1	1	2	3	3	3	-	1	1	2	2	-	-	-
Gray Malleable Iron	1	1	2	3	3	3	-	-	1	2	2	-	-	-
Maraging Steels	-	-	-	-	-	-	1	2	2	2	2	-	-	-
High Speed Steels	-	-	1	2	2	2	-	-	2	2	2	-	1	1
Tool Steels	-	-	1	1	1	1	-	1	1	2	2	-	1	1
High Manganese	-	-	1	1	2	2	-	2	2	2	2	1	1	1
Magnesium Alloys	-	-	-	-	-	-	1	1	2	2	2	2	2	2
Aluminum Alloys	-	-	-	-	-	-	1	2	2	2	2	2	3	3
Titanium Alloys	-	-	-	-	-	-	1	1	2	2	2	1	1	2
Pure Titanium	-	-	-	-	-	-	1	1	2	2	2	-	2	2
Copper Alloys	-	-	-	-	-	-	1	2	2	2	2	2	3	3
Brass & Bronze	-	-	-	-	-	-	1	2	2	2	2	2	3	3
“S” & “K” Monels	-	-	-	-	-	-	1	1	2	3	3	1	2	3
Zinc Alloys	-	-	-	-	-	-	1	1	2	3	3	1	2	3
Zirconium	-	-	-	-	-	-	1	2	2	2	2	1	1	2
Manganese	-	-	-	-	-	-	1	2	2	2	2	1	1	2
Plastic	-	-	-	-	-	-	1	1	1	1	1	2	2	3
Hi-Temp Alloys	-	-	-	-	-	-	1	2	2	2	3	-	2	2
Chilled Cast Iron	1	1	2	2	2	3	-	-	-	-	-	-	-	-
200-300 Stainless	-	-	1	1	1	1	1	1	2	2	3	-	2	2
Ultra-High Strength	-	-	2	2	2	2	-	1	1	1	1	-	-	-

1 = First Choice – the best possible selection

2 = Second Choice

3 = Third Choice

\- = Not Recommended

Troubleshooting

Excessive Edge Wear

(a) Increase chip load

(b) Reduce surface speed

(c) Check to see if the cut is climb or conventional milling. Climb milling is preferred.

(d) See if inserts are honed too heavily. Extra heavy hone will increase wear.

(e) Use a more wear-resistant grade.

Early Insert Breakage

(a) Check for rigidity of setup

(b) Make sure inserts are properly seated in pockets. One insert that protrudes could break and cause breakage of all inserts.

(c) The chip load may be too heavy

(d) Use tougher grade of carbide

Heat Checking

(a) Dry cutting will result in less heat checking than a marginal coolant setup.

(b) If a coolant flood is used, it should be an extra heavy flow from a nozzle with a large opening.

(c) Spray-mist type coolant would be preferred.

(d) Inserts should not be permitted to work too long, particularly negative rake inserts. The longer an insert cuts, the greater the potential for heat checking. Negative rake inserts that are severely heat checked usually cannot be turned over, as they tend to break more easily.

Excessive Cratering

Cratering is seldom a problem on milling operations. However, if the problem does exist it can be reduced by using a more wear-resistant grade of carbide.

Edge Chipping with Subsequent Breakage

There are numerous causes for insert chipping. Some are easily detected and corrected while others are more difficult.

(a) Make sure proper edge treatment is applied.

(b) Check for rigidity of setup

(c) Reduce chip load

(d) Increase speed

(e) Use a tougher grade of carbide.

Chip Clinging to Cutting Edge

Chips welding to the cutting edge are a common cause for rapid tool failure. The chip clings to the cutting edge as it leaves the cut and gets wedged between the edge and the work piece. Each time the insert enters the work piece, the chip is knocked off, carrying particles of carbide with it. Chip welding can be prevented by the following:

(a) Increasing cutting speed

(b) Substitute coated carbides

(c) Increase feed

(d) Increase rigidity of setup.