Technical Information

Choosing Cutter Diameter

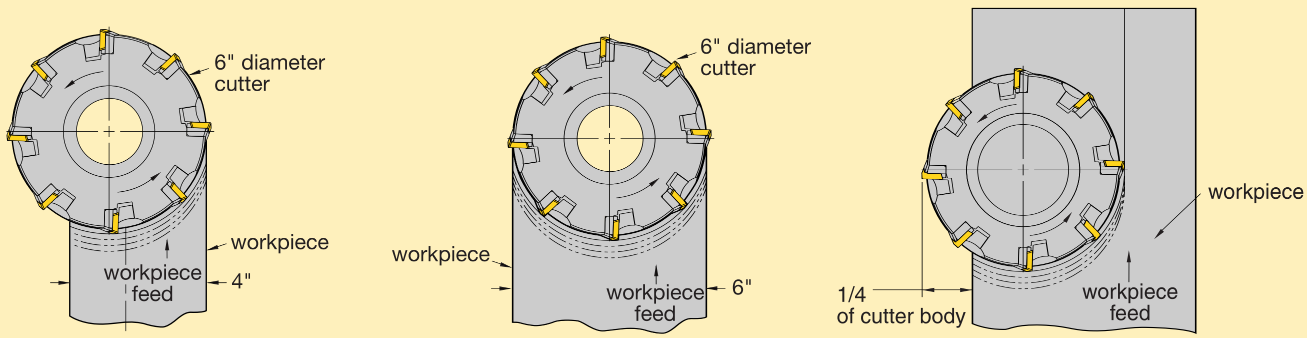
Workpiece dimensions determine the best face mill diameter to select.

Cutter to part width-of-cut ratio should be approximately 3:2 or 1 1/2 times the part width. For example, if the width of cut is four inches, choose a six-inch diameter cutter. If the width is extremely wide, select a cutter diameter that matches the spindle capacity and take multiple passes. For example, if the width of cut is 24 inches and the machine has a standard #50 taper spindle, you should use an eight-inch diameter cutter and take five passes, at slightly less than five inches per pass, or four passes at six inches per pass, depending on horsepower and rigidity.

An undesirable situation is when the cutter diameter is about equal to the width of cut. The chip being formed at the entrance and exit of the cut will be very thin. The thin chips formed cannot carry away heat as well as thicker chips, therefore the heat is transferred back into the insert causing premature edge failure. Workhardening is also more likely to occur in the entry and exit area.

When the proper cutter diameter is not available, proper cutter positioning will provide positive results.

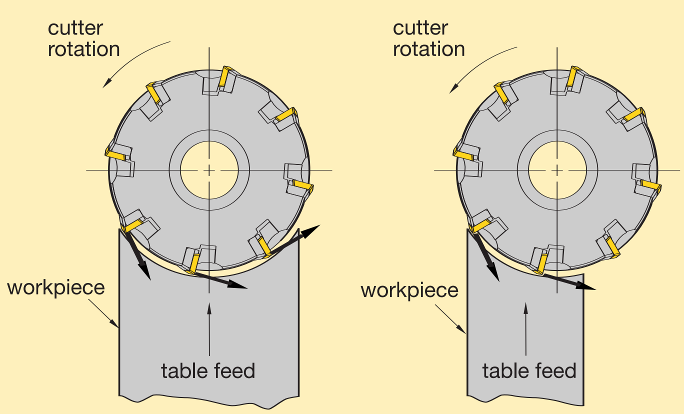
- Position cutter with approximately 1/4 of the cutter body outside the workpiece and make two passes.
- Produces negative angle of entry (desirable).
- Can result in longer tool life.



desirable **undesirable** **cutter position**

Cutter Positioning/Cutting Forces

The cutting forces are constantly changing as the inserts move through the cut. We should understand that in changing the position of the cutter in relation to the workpiece, we can re-direct the cutting forces. This is important to ensure a safe operation based on fixture design, workpiece design, and workpiece considerations.



Cutter Pitch

Pitch, or density, refers to the number of inserts in a cutter. Cutters can be classified as having either coarse, medium, or fine pitch. When designing a cutter, the engineer must take the depth of cut and feed per tooth into consideration. He then must provide the necessary chip clearance in the body so that the chip can pass without restricting its formation. For this reason, cutters designed for heavy metal removal have maximum chip clearance. This, therefore, restricts the number of inserts in the cutter, making it a coarse pitch cutter.

In medium pitch cutters, the chip clearance area in the body is usually slightly smaller than a coarse pitch cutter. And, in fine pitch cutters, the chip clearance is considerably less.

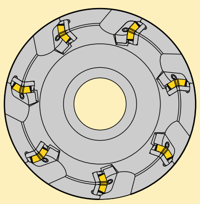
Coarse pitch is recommended for general purpose milling where adequate horsepower is available, and where maximum depth of cut is required.

Medium pitch is recommended when moderate feed per insert is required, and where it is more advantageous to have more than one insert in the cut. Medium pitch also reduces entry shock and cutting pressure while maintaining feed rates.

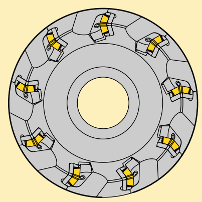
Fine pitch is ideal when milling a severely interrupted surface such as a manifold block. Fine pitch cutters are capable of higher inch per minute feed rates than medium or coarse pitch cutters. They also experience higher cutting forces and greater horsepower consumption than medium or coarse pitch cutters do.

Differential Pitch

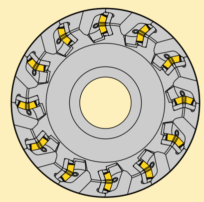
A cutter with unequally spaced inserts is a differential-pitch milling cutter. This configuration breaks up the harmonics that result from equally spaced inserts, greatly reducing the chance of vibration. Most cutter designs today use this design feature regardless of the cutter pitch.



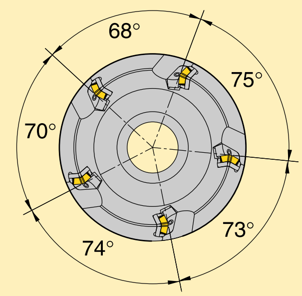
coarse pitch



medium pitch



fine pitch



differential pitch

Lead Angles/Cutting Forces on Workpiece and Fixturing

Cutting forces produced during the milling process are constantly changing as the insert moves through the cut. Understanding the relationship of these forces will help ensure safe operation by preventing workpiece movement during the cut. For example, fixture design and clamp positioning are determined by the cutting forces produced in milling. Equally important is an understanding of the effect lead angle has on cutting force direction, actual chip thickness, and tool life.

0° lead angle

advantages:

- When 90° shoulder is required
- Can be a problem solver on thin wall workpieces

disadvantages:

- Highest radial cutting forces
- High entry shock load
- Increased chance of burr on insert exit side of part

15° and 20° lead angle

advantages:

- For general milling applications and relatively rigid conditions
- Good relation of insert size and maximum depth of cut
- Reduced entry shock load

disadvantages:

- Higher radial forces can cause problems in weak machine/workpiece/fixture conditions

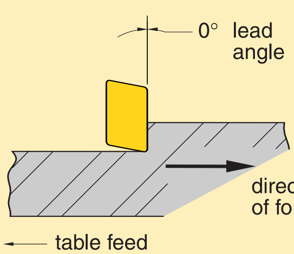
45° lead angle

advantages:

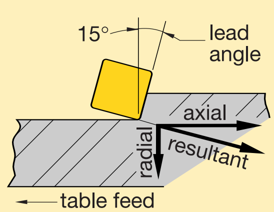
- Well balanced axial and radial cutting forces
- Less breakout on workpiece corner
- Entry shock minimized
- Less radial forces directed into spindle bearings
- Higher feed rates possible

disadvantages:

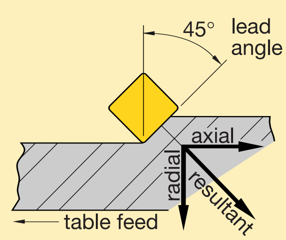
- Reduced maximum depth of cut due to lead angle
- Larger body diameter can cause fixture clearance problems



0° lead



15° and 20° lead

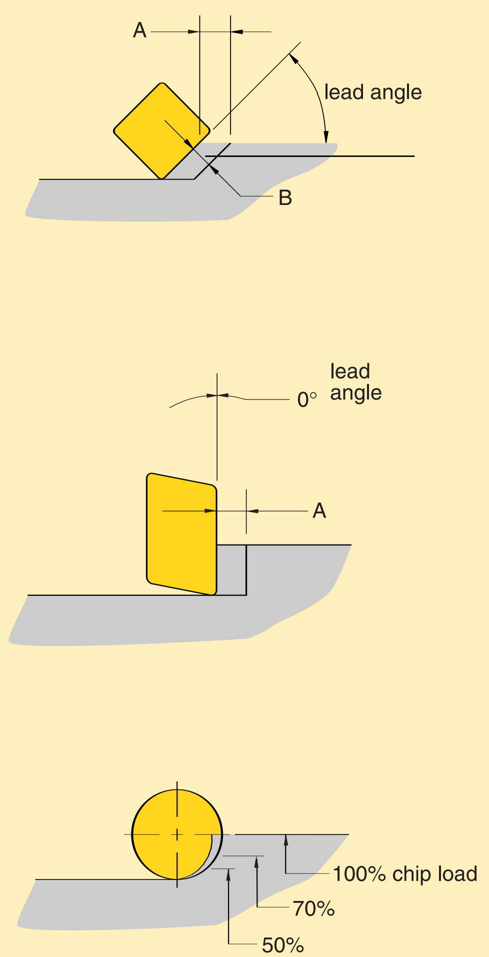


45° lead

Lead Angle/Chip Thickness

Chip thickness is affected by lead angle. The greater the lead angle, the thinner the chip will be since it's distributed over a greater length of the cutting edge. To achieve greater productivity and problem-free milling, use a lead angle cutter whenever possible.

lead angle	inch per tooth (feed)	actual chip thickness "B"
0°	A	A
15°	A	.96 x A
20°	A	.94 x A
30°	A	.86 x A
45°	A	.707 x A
example:		
0°	.010	.010
15°	.010	.0096
20°	.010	.0094
30°	.010	.0086
45°	.010	.0071



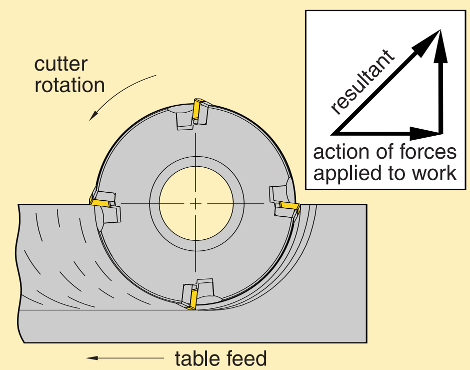
On round inserts, the chip load and lead angle vary with the depth of cut.

lead angle/chip thickness

Conventional Milling

For many years it was common practice to mill against the direction of the feed, due to the use of high-speed steel cutters and an absence of backlash-eliminating devices. The milling procedure became known as conventional, or up-milling.

In conventional milling, friction and rubbing occur as the insert enters into the cut, resulting in chip welding and heat dissipation into the insert and workpiece. Resultant forces in conventional milling are against the direction of the feed. Work-hardening is also likely to occur.

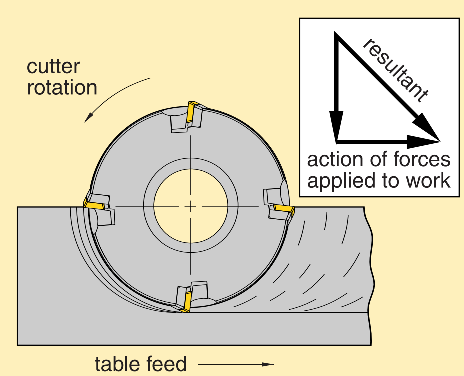


conventional milling

Climb Milling (preferred)

Climb milling is normally recommended. The insert enters the workpiece material with some chip load and produces a chip that thins as it exits the cut. This reduces the heat by dissipating it into the chip. Workhardening is minimized.

Climb milling forces tend to push the workpiece towards the fixture and in the direction of the feed. Climb milling is preferred over conventional milling in most situations.



climb milling