

Understanding Thread Mills

Any three axis mill that is capable of helical interpolation can be used for thread milling. Helical interpolation involves three axes moving simultaneously. Two axes, 'X' and 'Y', move in a circular motion while the 'Z' axis moves in a linear motion. For example, the path from point A to point B (see Figure 1) on the periphery of the cylinder combines a circular movement in the 'X-Y' plane with linear movement along the 'Z' axis. The 'X' and 'Y' circular motion will determine the diameter of the thread. The 'Z' axis linear motion will cut the pitch (or lead) of the thread.

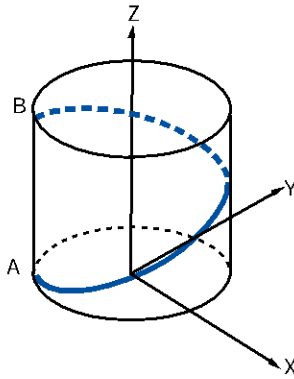


Figure 1

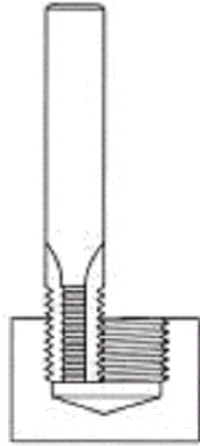


Figure 2

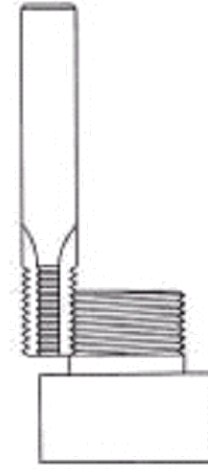


Figure 3

How to Order Thread Mills

Thread mills must completely enter the minor thread diameter before cutting the internal thread (see Figure 2). Thus our catalog lists the smallest internal thread that each thread mill can produce. The same thread mill can also produce any larger size internal thread of that same pitch. Also, for small sizes, it is best to use our short series with the reduced length of cut if possible.

All of the straight flute thread mills are for internal threads only. All of the staggered tooth thread mills will cut both the internal and external threads. The helical thread mills over 0.187 diameter will also cut both internal and external threads.

Staggered tooth thread mills have every other tooth removed in a staggered pattern. As the tool rotates the adjacent flute fills in for the tooth that was removed. This helps to reduce side cutting pressure, thus reducing chatter. This can be extremely beneficial in small external sizes and for set-ups that lack rigidity.

Helical thread mills are also designed to reduce side cutting pressure by distributing the cutting pressure along a helical flute. Although these tools cost slightly more, their high performance design allows for less chatter and higher feed rates.

How to Use Thread Mills

To produce internal threads, drill the minor thread diameter to its appropriate size. Then, position the thread mill to the required depth. Next, mill either the 'X' or 'Y' axis to the required thread pitch diameter. With small sizes and with difficult to cut material, it may be necessary to remove material in several passes. It is always best to "arc in" and "arc out" when thread milling. Any "arc in" and "arc out" movements must have a corresponding 'Z' axis motion during the 'X-Y' circular moves. For example, if the "arc in" is over 90 degrees, the 'Z' axis departure must be $\frac{1}{4}$ of the thread pitch. (90 degrees is $\frac{1}{4}$ of a circle). An internal right-hand thread is produced by orbiting in a counterclockwise direction while bringing the 'Z' axis up one pitch per 360 degrees.

A left-hand internal thread is produced by orbiting in a clockwise direction while bringing the 'Z' axis up one pitch per 360 degrees. The entire process can be achieved by interpolation in a downward direction and reversing the orbit direction.

External threads (see Figure 3) must have the major diameter milled to size before a staggered tooth thread mill is used. Helical flute thread mills are crest cutting on external threads. Right-hand external threads are cut by interpolating down and in a clockwise direction. The same threads can be cut by interpolation up and changing the orbit direction.

NPT threads are usually produced while interpolating the tool in a downward direction. Since these tools are crest cutting, it is not absolutely necessary to ream the internal minor diameter or mill the external diameter to size. However, it is highly advisable to do so since the tools will have much less material to remove. If the tool is to be interpolated in an upward direction, spiral interpolation must be used.

The same surface feet per minute can be used for thread mills as for end mills of the same size. The feed rate must be slower, however, since thread milling often involves unfavorable length-to-diameter ratios. Also, keep in mind that the thread mills have more surface area contact than an end mill of equal length. Most CNC mills are programmed in inches per minute which is applied at the centerline of the spindle. In internal applications, the outside diameter of the tool will be traveling faster than the centerline of the tool. The reverse is true for external applications. It is best to start out conservatively with feed rates and the number of passes required and adjust upward per good machining practice.

