OCTOBER 2002 / VOLUME 54 / NUMBER 10

BY JIM ROWE

The Lowdown on Laydown Inserts

Understanding the benefits of a laydown threading system.

In contrast to "on-edge" or "top-notch" inserts, "laydown" threading inserts sit in the tool body like standard turning inserts. The advantage of this is that the holder's pocket walls and seat firmly secure the insert.

All the benefits of using laydown-threading tools often aren't explained to shops that turn threads. Most manufacturers that offer the tools focus on explaining the concept of the "full-profile" style of laydown insert, which creates a thread without sharp edges and with great concentricity between the major and pitch diameters—a nice benefit. Or they promote the "partial-profile" style that enables you to produce 8 to 48 threads per inch.

But few tool manufacturers make an effort to clearly explain the laydown tool's most important feature: That it lets the end user adjust the insert in angular increments to achieve the proper helix angle, which prevents the tool from rubbing on either side of the cutting edges. This is achieved by having an angled seat—known as an anvil and available in 1° increments—beneath the insert (Figure 1). A threading tool with the proper clearance lasts longer and produces a higher-quality thread with a truer shape.

Most tool manufacturers' catalogs contain a section that shows the many holders and inserts available for laydown-threading systems. Somewhere toward the end of the technical information, there are often confusing charts and formulas that are included to try and explain the helix angle and the proper anvil for these laydown tools.

This article explains the theories, characteristics and benefits of laydown-threading tools to help shops improve their thread quality and threadmaking productivity.

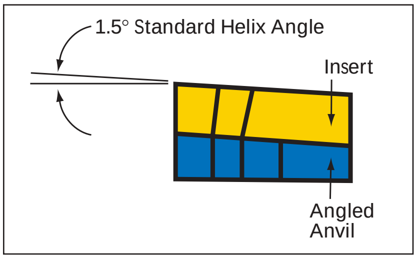
Angles and Anvils

A print specifies the thread's major diameter and thread pitch, along with a class designation. This combination of thread diameter and pitch determine the thread's helix angle.

Technically, a thread's helix, or lead, angle is the distance a threaded part moves axially (with respect to a fixed mating part) in one complete revolution. The lead angle is equal to the pitch multiplied by the number of thread starts. This angle is evident by how much the thread appears to lean in one direction. Coarse threads have a rather steep angle, and fine threads have an angle that's less steep.

The laydown threading tool's anvil is an angled seat located underneath the insert. The anvil that comes with a tool body normally has a 1.5° angle, which is appropriate for making many types of threads.

However, any thread with a major diameter smaller than 5⁄16" needs to be made with a tool having an anvil with a helix angle greater than 1.5° (Table 1). If the thread requires an anvil with a greater angle than what you selected, the tool could rub the side of the thread. This can reduce tool life and create poor-quality threads. Most manufacturers designate anvils as having ±1° to 2° above the 1.5° standard. In addition, the insert's cutting edges need to be properly aligned according to the distance



All images supplied by VNE Corp.

Figure 1: An anvil allows the end user to adjust the angle the insert is set at, which prevents the tool from rubbing.

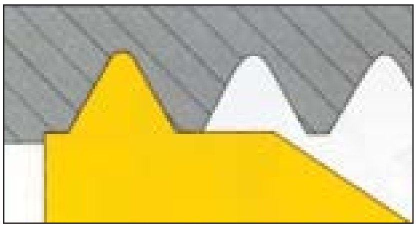


Figure 2a, 2b: The full-profile insert (top) is designed to cut a specific thread pitch, while the partial-profile style can cut threads with pitches from 8 to 48 tpi.

per revolution that the pitch dictates.

Refer to the tool manufacturer to be sure you purchased a tool with the correct anvil. Also check the differences for internal and left-hand threads, as there are general rules for these types of threads as well.

Ground vs. Molded

There is always a choice with laydown threading systems of having molded or ground insert edges. The ground edges provide a better cutting action and impart a finer surface finish when making threads in materials like stainless steel, titanium and aluminum. However, ground edges are typically combined with a ground chipbreaker, which doesn't provide the best chip control. This can cause a big nest of chips to gather above the tool and wrap around it while it's cutting, resulting in premature tool failure and poor thread quality.

Some time ago, tool manufacturers figured the answer was to mold a chipbreaker into the top of the insert to provide better chip control during machining—and it does. However, inserts with a molded chipbreaker typically come with molded cutting edges, so the cutting action and surface finish aren't as good. Therefore, a compromise had to be made.

Some manufacturers produce laydown inserts with molded chipbreakers and precision-ground cutting edges. These inserts offer effective chip control and impart smooth thread surfaces. Moreover, if they're TiAlN-coated carbide inserts, the cutting speed can be increased and the number of passes reduced.

System Benefits

There are a number of considerations when choosing a laydown threading system. If you have many different thread pitches to cut every day and are not in a production environment, the partial-profile insert might be the best choice (Figure 2b). By applying one partial-profile insert, you can cut from 8 to 48 tpi and reduce inventory costs and tool-change time vs. a full-profile insert. The partial-profile insert combined with the correct anvil and a molded chipbreaker can generate great results. Since the partial-profile insert does not cut the major diameter, there will be a sharp edge on the top of the thread.

If you are in a production environment and want to produce one specific thread pitch continually, choose the full-profile insert (Figure 2a). This insert is ground for a specific thread pitch.

The profile that is ground or molded has the exact dimensions to machine the major, pitch and minor diameters. By turning the major diameter a few thousandths of an inch over the required major diameter, the full-profile insert skims the major diameter as it finishes the other dimensions, resulting in great concentricity between all diameters. You need to purchase inserts for every thread pitch required, but the return on investment will be tremendous in the long run.

Where to Start

The programming of threads on CNC lathes is simplified by the threading cycles offered by control manufacturers. These cycles have a reference point as a default. Many times, with coated carbide grades that are able to run at a high cutting speed, the Z-axis reference points are too close to where the tool starts cutting for the speed of the spindle that is turning. When the Z-axis reference distance is too close, the turret's huge mass will not allow it to synchronize the feed until after the second or third thread is cut.

Realize that if you are turning at 2,000 rpm and cutting 20 tpi (0.050 ipr), the turret must travel at 100 ipm from a dead stop. Obviously, the coarser the thread, the faster the turret must travel.

If the first part of the thread has a poor "V" effect, you might slow the spindle speed to fix the problem. But

Thread	Helix Angle	Thread	Helix Angle
5-36	3.5°	M1.6 × 0.35	2.5°
5-40	3.5°	M2 × 0.4	2.5°
5-44	3.5°	M2.5 × 0.45	2.5°
6-32	3.5°	M3 × 0.5	2.5°
6-36	3.5°	M3.5 × 0.6	3.5°
6-40	3.5°	M4 × 0.7	3.5°
7-32	3.5°	M5 × 0.8	3.5°
8-30	3.5°	M6 × 1	3.5°
8-32	3.5°	M8 × 1.25	2.5°
8-36	3.5°	M8 × 1	2.5°
8-40	2.5°	M10 × 1.5	2.5°
9-32	3.5°	M10 × 1.25	2.5°
10-24	3.5°	M10 × 0.75	1.5°
10-28	3.5°	M12 × 1.75	2.5°
10-30	3.5°	M12 × 1.5	2.5°
10-32	2.5°	M12 × 1.25	1.5°
12-24	3.5°	M12 × 1	1.5°
12-28	2.5°	M14 × 2	2.5°
12-32	2.5°	M14 × 1.5	1.5°
¼-20	3.5°	M15 × 1	1.5°
¼-28	2.5°	M16 × 2	2.5°
5/16-18	3.5°	M16 × 1.5	1.5°
5/16-24	2.5°	M17 × 1	1.5°
⅜-16	2.5°	M18 × 1.5	1.5°
⅜-24	1.5°	M20 × 2.5	2.5°
7/16-14	2.5°	M20 × 1.5	1.5°
7/16-20	1.5°	M20 × 1	1.5°
½-13	2.5°	M22 × 2.5	2.5°
½-20	1.5°	M22 × 1.5	1.5°
9/16-12	2.5°	M24 × 3	2.5°
9/16-18	1.5°	M25 × 1.5	1.5°
⅝-11	2.5°	M27 × 3	2.5°
⅝-18	1.5°	M27 × 2	1.5°
¾-10	2.5°	M30 × 3.5	2.5°
¾-16	1.5°	M30 × 2	1.5°
⅞-9	2.5°	M30 × 1.5	1.5°
⅞-14	1.5°	M33 × 2	1.5°
1-8	2.5°	M35 × 1.5	1.5°
1-12	1.5°	M36 × 4	2.5°
1¼-7	1.5°	M36 × 2	1.5°

Table 1: The helix angles needed to cut the most popular threads.

		THREADS PER INCH									
		8	**10**	**12**	**14**	**16**	**18**	**20**	**24**	**28**	**32**
RPM	**2,500**	0.781	0.625	0.521	0.446	0.391	0.347	0.313	0.260	0.223	0.195
	2,250	0.703	0.563	0.469	0.402	0.352	0.313	0.281	0.234	0.201	0.176
	2,000	0.625	0.500	0.417	0.357	0.313	0.278	0.250	0.208	0.179	0.156
	1,750	0.547	0.438	0.365	0.313	0.273	0.243	0.219	0.182	0.156	0.137
	1,500	0.469	0.375	0.313	0.268	0.234	0.208	0.188	0.156	0.134	0.117
	1,250	0.391	0.313	0.260	0.223	0.195	0.174	0.156	0.130	0.112	0.098
	1,000	0.313	0.250	0.208	0.179	0.156	0.139	0.125	0.104	0.089	0.078
	750	0.234	0.188	0.156	0.134	0.117	0.104	0.094	0.078	0.067	0.059
	500	0.156	0.125	0.104	0.089	0.078	0.069	0.063	0.052	0.045	0.039
	250	0.078	0.063	0.052	0.045	0.039	0.035	0.031	0.026	0.022	0.020

Table 2: The Z-axis reference-point distance for laydown threading can be determined by matching the thread pitch (tpi) with the spindle speed (rpm).

then the tool may not be running at its appropriate speed, preventing the full benefits of the laydown system from being realized. To realize the system's full benefits, adjust the distance and allow the tool to cut at the proper sfm. You'll extend tool life.

When choosing a laydown threading system, determine the anvils needed to produce the desired threads, select ground or molded cutting edges and chipbreakers based on the application and decide whether a full- or partial-profile insert works best. In addition, find out what the Z-axis reference point should be (Table 2). With the correct combination, you should see great results when turning threads with laydown threading inserts.

About the Author

Jim Rowe is a cutting tool application salesman for a Midwest distributor.