JANUARY 2005 / VOLUME 57 / NUMBER 1

➤BY ALAN RICHTER, MANAGING EDITOR

KNOW YOUR LIMITS

Understanding pitch-diameter limits and class of thread fit helps in specifying the appropriate tap for the job.


Emuge

The required pitch-diameter limit varies slightly from one tap style to the next.

Determining whether a tapped hole is within spec is a straightforward process. "As long as the GO gage goes into the hole, and as long as the NO-GO gage doesn't go into the hole, it's acceptable," said Alan Shepherd, technical director for tapmaker Emuge Corp., Northborough, Mass. "The NO-GO gage, of course, being the larger gage."

"Similarly, the acceptability of a male part with an external thread is also determined by a corresponding GO thread ring gage," said Dan Gajdosik, engineering manager for tapmaker Besly Products Corp., South Beloit, Ill.

Determining which tap will produce acceptable threaded holes while maximizing tool life is a little trickier, but not that complicated once the basics are understood. These include pitch, pitch diameter, class of thread fit and tap pitch-diameter limits.

Pitch is the distance from a point on a thread to a corresponding point on the adjacent thread, measured parallel to the axis of the thread. The pitch in inches is equal to 1.0" divided by the number of threads per inch. For example, the pitch of a 6-40 UNC tap is 1 divided by 40, which equals 0.025".

Picturing an imaginary cylinder helps to comprehend the basic pitch diameter of a screw thread. On a straight screw thread, the surface on an imaginary cylinder's diameter would pass through the threads at such points as to make equal the width of the threads and thread groove intersected by the cylinder's surface (Figure 1).

Pitch diameter is a major attribute, or characteristic, of a tap, but it is just one attribute of many. Shepherd pointed out that there are other tap attributes, such as the flutes, relief characteristics and lip heights on the chamfers, as well as the application, toolholder and workpiece material characteristics, that impact how a tap performs. "Pitch diameter does not by itself control the end result when tapping a thread."

Class of Fit

According to Besly Products, all measurements must have a controlling point, or base, from which to start. For a thread screw, this starting point is called the basic, or theoretically correct, size. While it is impossible in practice to form a screw thread to its basic size, it is possible and practical to establish limits that the deviation must not exceed. These are called the maximum and minimum limits. The difference between the maximum and minimum limits is the tolerance.

The classes of thread fit are distinguished from each other by the level of tolerance and allowance between the tapped hole and the screw or bolt, noted

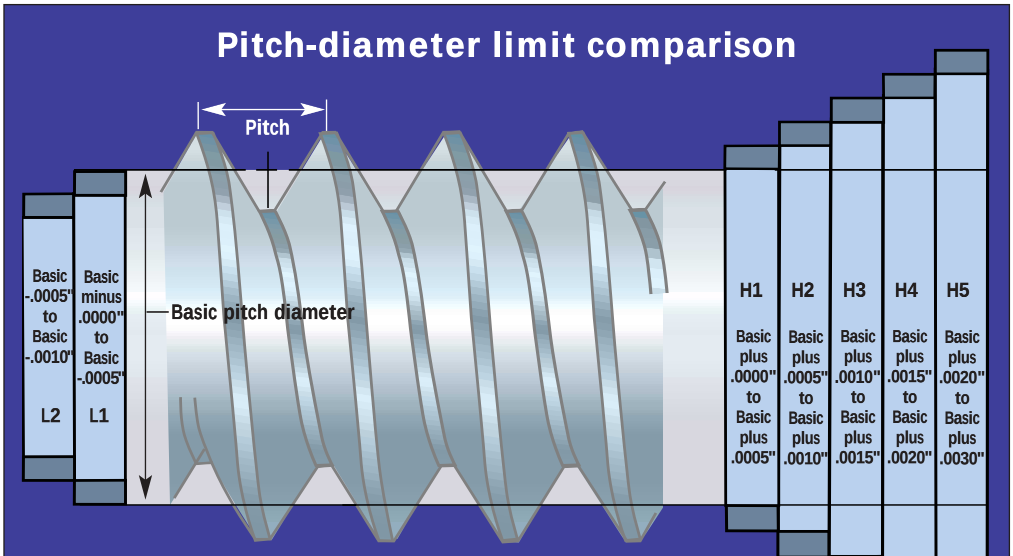


Figure 1: H-limits are used to properly size a tap for the threaded hole to be produced. H-limits are a sequence of steps in 0.0005" increments beginning at the minimum size limit of the part, starting with H1.

Thomas W. McClure, vice president of sales for tap manufacturer Balax Inc., North Lake, Wis. There are three classes of fit for threaded assemblies. Class 1 has the loosest tolerance and covers threaded parts for which quick and easy assembly is desired. This class generally applies to do-it-yourself-type nuts and bolts.

McClure said Class 2 has an intermediate tolerance to minimize any galling and seizure when assembling threaded parts. "It also accommodates, to a limited extent, plating, finishes and coatings," he added.

The tightest-tolerance class is Class 3, and it applies to threaded assemblies for which the closeness of fit and the accuracy of the threads' lead and angle are critical. "These threads are obtained consistently only by use of high-quality production equipment supported by an efficient system of gaging and inspection," McClure said.

The class of thread fit includes the letter A for external threads and the letter B for internal ones.

"Most internal threads have a Class 2B thread," said Shepherd. "Aerospace products, safety-critical products, weapon systems and rotating parts often have a Class 3B fit. It costs more money to hold a closer class of fit, so you don't put it where it's not needed. You would want a closer tolerance on, say, the connecting rod on your car than the bolt that's holding the bumper."

Besly's Gajdosik emphasized that only when both members of a threaded assembly fall within the desired class of fit can the proper fit be assured.

Properly Sizing a Tap

Once the class of thread fit has been determined, an H- or L-limit for screw machine and fractional-inch taps is specified by the manufacturing engineer to create a thread within the minimum and maximum limits for the chosen class of fit. For metric taps, a D-limit is specified.

"H" means the pitch diameter is on the high side of basic pitch diameter. H-limits for taps provide a range of pitch-diameter limits larger than the basic pitch diameter, with a maximum limit in increments of 0.0005" for each H-limit, Shepherd explained. For example, H1 equals a maximum of 0.0005", or basic to +0.0005", H2 equals a maximum of 0.0010", or basic +0.0005" to +0.0010", and so on.

On the other end of the spectrum, a tap with an L-limit, or low side of basic, would have a range of pitch-diameter limits smaller than the basic pitch diameter. For example, an L1 tap is below the basic pitch diameter by up to 0.0005". A tap with an L-limit is for a special application where an interference fit for the threaded assembly is desired, often creating a permanent assembly, Gajdosik noted. "It's when you want to be sure the fit remains forever." Normally, L-limits have no connection with 2B and 3B classes of fit.

Shepherd added that for taps 1.0" in diameter and smaller, a manufacturing tolerance of -0.0005" is subtracted from the maximum value of each H-limit. Therefore, H1's minimum/maximum value is 0.0005"/0.0000", H2's minimum/maximum value is 0.0010"/0.0005", and so on. Taps larger than 1.0" have a looser tolerance on the pitch diameter, but the maximum limit value will always be in increments of 0.0005".

"The H-limit of a tap determines that tap's physical pitch diameter in relation to the GO gage for a thread," explained McClure. "Each H number is equal to a multiple of 0.0005" over the GO gage."

D-limits for metric threads function in the same manner, except the values are calculated in millimeters, where each D-limit increment is 0.013mm, which is slightly larger than 0.0005".

Adding to the alphabet soup, the letters GH before the limit number designate the tap has been ground high over the basic pitch diameter rather than ground low (GL), which is below the basic size, Gajdosik said. "In general, tolerances for internal threads are above basic and external threads are below basic to allow for assembly of the parts."

Gajdosik added that although tapmakers grind taps to precise tolerances under closely controlled manufacturing processes and guarantee the accuracy of individual elements, there is always the possibility of the presence of unknown factors that can be detrimental to tap performance. These factors include the chemical and mechanical properties of the workpiece material, spindle runout, workhardening effects, clamping pressure causing distortion on a thin-walled part, thread depth and hole preparation. "The tap manufacturer, therefore, is not able to guarantee the size of the tapped hole," he said.

Selecting a Limit

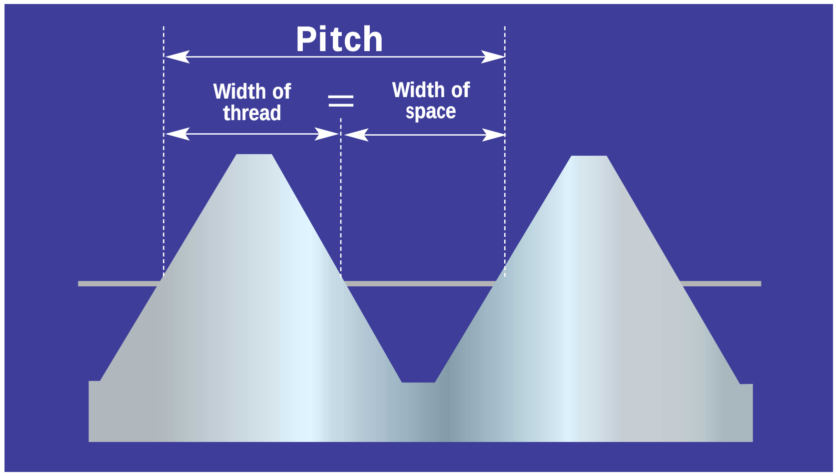
If the threaded hole closes in because of shrinkage from heat treatment or springs back slightly because of the metal's memory or there is buildup on the threads from plating, no formula exists to determine the exact pitch-diameter limit required. This limit compensates for the changes in lead variation, flank angles and size. Generally, H- or other limits are determined by trial-and-error field testing, although guidelines and published charts exist to help determine the proper limit. Some tap manufacturers also offer software that provides recommendations.

For instance, parts that are heat-treated before being tapped require a tap with an H-limit higher than when tapping parts that are not heat-treated, Gajdosik said. This is because the threads may distort and the part may shrink 0.0015" to 0.0025", or three to five H-limits, depending on the material. He added that higher-than-usual H-limits are also recommended when tapping brass, copper and titanium, which tend to spring back after the tap is withdrawn.

In cases where the part will be plated after tapping, Gajdosik said the rule of thumb is to quadruple the plating thickness and add that amount to the GO and NO-GO pitch diameters. Then select a tap with a pitch diameter between those two values.

"Normally, you select an H-limit that is midway between your GO/NO-GO gage limits," Emuge's Shepherd said. For example, a ½-13 UNC-2B GO/NO-GO gage with a GO diameter of 0.4500" and a NO-GO diameter of 0.4565" has a tolerance of 0.0065". When 0.0065" is divided by two to achieve the midway point, the result is 0.003", so an H6 tap that is 0.0030" greater than the basic pitch diameter would be chosen.

"This is an area where much controversy comes into play," Shepherd noted. "A customer may have an H3 tap in inventory and, for sure, that tap would work in this application as well."



Pitch is the distance from a point on a thread to a corresponding point on the adjacent thread, measured parallel to the axis of the thread.

He added that the slightly smaller tap would produce a threaded hole that would gage correctly, but may not have the same life as the larger one. Because a tap manufacturer might manufacture more, say, ¼-20 UNC H3 taps than ¼-20 UNC taps with other H-limits, the H3 tap would be less expensive. However, an end user would probably get significantly more life out of a slightly larger H5 tap. "That, to me, is one of the areas people really miss the boat on," emphasized Shepherd. "As a society, we look for what deal we can get and sometimes the deal we get really isn't worth it."

On the other hand, sometimes less can be more. "It is not always the most expensive tap that will do the job best," said Gajdosik. "There are many instances where special tapping jobs were solved with inexpensive standard taps."

Instead of focusing on the midway point of the GO/NO-GO gage's tolerance, others follow the "40 percent rule" to handle the widest variety of tapping conditions. "For cutting taps, a rule of thumb is to use a tap H number that is 40 percent of the range between the GO and NO-GO limits of the class of thread fit being produced," said Balax' McClure. "For example, if the tolerance between the GO gage and NO-GO gage is 0.005", a tap at 40 percent would be made 0.002" over the GO gage, which makes the tap an H4, or 0.002" divided by 0.0005"."

Gajdosik added that the 40 percent rule provides a safety factor. With the rule, "the tap will produce threads where the GO gage will enter freely and the NO-GO will only enter a maximum number of turns before resistance is felt."

Troubleshooting

When oversize tapped holes are a result of galling, Gajdosik pointed out that an opportunity for problem-solving exists. He indicated that numerous questions need to be answered to find the cause.

■ Is the recommended H-limit being used?

■ Is a tapping fluid being applied, is it appropriate for the workpiece material and is it being applied correctly to flush the chips out of the tapped hole?

■ Is the tapping speed correct for the type of material and depth of the hole being tapped?

■ Is a surface treatment to the tap recommended for the material?

■ Is the hole being tapped the correct diameter?

■ Is there a misalignment or spindle runout condition?

■ If the part is thin-walled, are the threads are bulging on the OD?

In those rare instances where there is no reasonable explanation for the problem's root cause, Gajdosik recommends reducing the tap's H-limit closer to the GO-gage's pitch diameter, even though a special tap may be required.

Understanding pitch-diameter limits is helpful for specifying the best tap for the job but not essential. As it turns out, pitch-diameter H-limits for taps are only specified in North America. “If you go to the rest of the world, be it Europe, Asia, South America or Africa, they don’t use them,” Shepherd said. “They go specifically for the class of thread fit and the tap is designed to produce that class of thread fit.”

He noted that Emuge marks the class of thread fit, such as 2B or 3B, for inch sizes instead of the H-limits on its taps. “These taps are designed for material-specific applications and the pitch diameter is an attribute that is designed for the material-specific cutting characteristics and the style of tap,” Shepherd explained. “This provides the optimal tap life and hole-size control. The reason we don’t put the H-limit on is it would confuse people immensely.”

But don’t expect these lettered limits to go by the wayside anytime soon. “As a matter of fact,” Shepherd said, “ASME B94.9 is in the process of revising the standard and it will stay with the H-limits system as we know it.”

The following companies contributed to this report:

Balax Inc.
(800) 886-1398
www.balax.com

Besly Products Corp.
(800) 435-2965
www.besly.com

Emuge Corp.
(800) 323-3013
www.emuge.com

CUTTING TOOL ENGINEERING Magazine is protected under U.S. and international copyright laws.
Before reproducing anything from this Web site, call the Copyright Clearance Center Inc. at (978) 750-8400.