- Thread Specifications -

Considerations when adding threads to a design.

Issue #3— Resource Library

Date: 1/1/2024

Special points of interest:

- Threading is a subject with many tangents requiring careful examination
- Learn about the ASME B1.3 Thread Inspection methods
- Higbee—Blunt Start threads
- Thread Insert Offerings
- Have you ever used Spiralock threads?
- Options for plating threads

Written by:

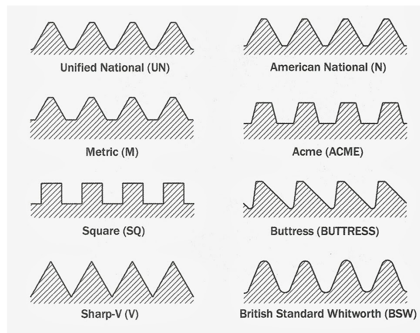
Jon Bancroft

Threads: One of the Deepest Rabbit Holes

Threads are supposed to be simple things. A simple machine, the inclined plane, wrapped helically around a cylinder. But once you start asking questions, threads become a really deep rabbit hole, with many tangential paths spiraling downward.

When in doubt, it is best to return to the foundational standards. Whether they be the ISO and ASME standards, or the Machinery's Handbook. At the end of the day, these are the governing rules for thread making and inspecting. But ultimately it is a question of what the customer wants and what the machine shop can offer that drive the conversation.

There are many different thread forms available, but in the metric and imperial systems, governed by ISO and ASME respectively, the most common is a simple 60° "Vee" groove. For simplicity we will focus on that form. And for the brevity of the article we will stick with the Imperial system, though almost everything is applicable in the metric system.



Here is an assortment of some commonly used thread forms.

Depending on the application there may be various materials being threaded, parts being threaded on different machines, using different tools, and being gaged in different ways. Which at the end of the day all go back to the standards.

Thread Tolerancing—Simple Version

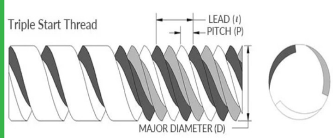
In the Imperial system, using the ASME Y14.5 standard, threads are specified using a simple one line callout on the blueprint. This line of text contains all of the necessary information an engineer should convey to the machinist who is producing the threads. A sample thread callout for a 1/4" bolt with 20 Threads Per Inch usually looks something like…

.250 - 20 UNC 2A

Where the diameter is called out first (Imperial uses decimals with no leading zero), then the thread pitch in T.P.I., then the governing UN standard Course/Fine etc., and then a number and letter combination. Where 1, 2, &3 are levels of precision, and where A or B are "external" or "internal" respectively.

There are extra optional arguments which can be added to the callout too.

Did you know? - The Full ASME Thread Tolerancing Callout



A Multi Start Thread

Source: [Harvey Performance Company]()

For the majority of applications the thread callout shown on Page #1 is more than adequate. However, did you know there is more to the ASME standard thread callout?

Our example was…
.250 - 20 UNC 2A

Where all imperial screw threads (except for pipes), whether they are Number series or Fractional (i.e. #6-32 & 3/8"-16) are represented with a decimal diameter, no leading zero. And then the usual callouts.

But a more full form callout can be shown as…
.250 - 20 UNC 2A R.H. (21)

The additional information here is the "R.H." and the "(21)". The R.H. is reserved for various thread modifiers, such as right/left handed threads, or multi start thread info.

The (21) represents the thread inspection method, see ASME B1.3. Where there are three options 21, 22, & 23. If "21", then a simple Go/No-Go check and basic Major/Minor Diameters. If "22" then more of the thread form needs verified quantitatively. And lastly "23" is extremely stringent and requires measurements of virtually every aspect of the thread.

"Note that for greater length to diameter ratios of thread engagement there is an allowed bonus tolerance"

Checking Threaded Holes—Nuts

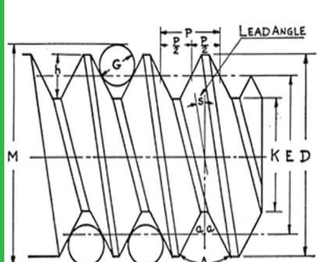
After a machinist produces a threaded hole they must check a number of things. Threaded holes are relatively easy to check if you have the right tools.

The first thing to check would be the minor diameter. The UN standard gives a formula for evaluating the minor diameter tolerance, or it can be found in the Machinery's Handbook tables. It is not governed by title block tolerances.

Note that for greater length to diameter ratios of thread engagement there is an allowed bonus tolerance to reduce the "thread percentage", thereby making threads easier to cut long distances.

Checking pitch diameter and major diameter is most easily accomplished with a Pass/Fail Thread Plug Gage. Where you need specific gages for every class of thread tolerance.

Checking Threaded Shafts—Bolts



Three-Wire method of pitch diameter measurement.
Source: [Thread Check Inc.]()

Inspecting external threads can be more challenging than internal threads. Namely, the inspection of pitch diameter.

While verifying the major diameter is relatively easy, checking the pitch diameter can be expensive. Since the variety of external threads and their tolerance classes is usually much broader than internal, there is either a need for a far larger quantity of thread "ring gages" or for alternative methods.

We use two redundant methods to verify our machined pitch diameters. We use "thread wires" as well as pitch diameter micrometers. Giving us two reliable ways to confirm the thread we cut meets specifications. Ring gages are available for common sizes and tolerances, but are expensive and take up much space.

UNJ Standard

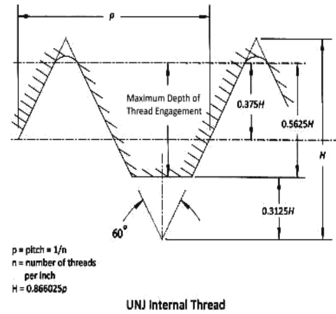
While the Unified National Standard governs almost every thread seen in machining, there are plenty of special standards. One of the more popular derivatives is the UNJ standard.

Functionally it is extremely similar to the UN standard. With the explicit intention of making more strong and more reliable threads, commonly seen in aerospace.

At the root of the differences is the fact that Internal threads have their minor diameter tolerance band shifted in the positive direction, and have a tight control placed on the form of the internal thread.

Whereas the External thread also has a tighter control on it's form, with the difference that it's root fillet must be larger than the ordinary UN standard.

The extra minor diameter in the "nut" allows for passage of the larger root fillet on the "bolt". This increased root fillet along with the tighter form tolerances yields a strong and reliable thread.



Note the additional root fillet in the UNJ standard.

Source: Gage Crib Worldwide

Tapping Methods

The most common form of threaded hole production in machine shops is tapping.

Tapping requires a pilot hole of the minor diameter on the workpiece. The hole should be straight and within tolerance of the standard. If a blind hole, then the depth should be dimensioned. Drill tip angles may be specified if needed, and optional thread relief grooves may be added. The hole should be chamfered to a minimum of the major diameter, typically 30°-45°.

Tapping is usually a cutting process, but can also be a forming process. Where "Cut Taps" remove material from the minor diameter out to the major. And where "Form Taps" actually require an enlarged minor diameter to begin with, and then during their advancement they exert pressure which flows the remaining ductile material inwards to shrink the minor diameter down to specification.

"Form Taps actually require an enlarged minor diameter to begin with"

Thread Milling

Thread milling is an alternative to tapping. Tapping is very fast and efficient, however it can also be risky. A broken tap may entirely scrap an otherwise finished workpiece. Thread milling alleviates this, as the tool is not "prisoner of it's own hole", as a tap is.

Thread milling also enables adjustment of fit tolerance, a finer surface finish, and the ability to machine nearly any pitch and diameter combination imaginable.

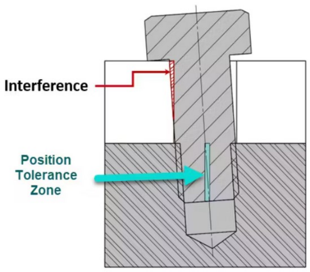
Thread milling is also a great way to cut large threads on lighter-duty machinery, where the cutting forces of thread milling are typically less than a similarly sized end mill.

Thread milling also gives the ability to cut custom threads, tapered threads, multi-start threads, etc. because it is fully CNC programmable.

Thread Mill Examples

Source: Carmex

Projected True Position Tolerance



Note the interference created by the off axis thread.
Source: GD&T Basics

Somewhat frequently we see the GD&T symbol for Projected Tolerance Zone applied to threaded holes. This can be helpful to know for making manufacturing decisions.

Essentially, the Projected Tolerance Zone is a continuation of the True Position tolerance up from the part surface to an imaginary plane above. Simulating the axis of the mating fastener in the real world application and it's fit within the adjoining part.

When we CNC machine most holes, they are drilled and threaded at the same time, and therefore are usually very perpendicular to their reference surface. However if we were working with a casting or other irregular part we have to be more careful in establishing our datums in machining and in inspection.

What is "Min. Full Thread"

"Care should be given to constrain the depth of a tap drill in blind hole applications"

Beyond the actual thread designation, there will also be a requirement to specify the length of thread engagement needed.

While specifying a depth for a tapped hole or a length of thread along a bolt are all that is required, sometimes this adds unnecessary cost to the parts.

Typically for a threaded hole or a shaft, all that functionally matters is that the threaded parts assemble with the proper amount of engagement.

Many designers use the technically not supported "Min. Full Thread" callout to imply a hole or shaft to have X.XX" of full thread forms, plus tolerance only.

While not supported in the standard, this is usually harmless, though care should be given to constrain the depth of a tap drill in blind hole applications.

This tip saves the extra machining & inspection effort to produce a precise exact thread depth to title block +/- tolerances.

When to use a Thread Insert

Assorted Screw Thread Inserts
Source: Stanley Engineered Fastening

Many people only consider using thread inserts as a repair tool. On the contrary, we mostly see them used on brand new parts. So why would you install a thread insert into an otherwise brand new component?

One reason is strength. We see this a lot in aluminum or copper parts where engineers specify stainless steel screw thread inserts.

Another reason we see this is for reducing galling between the fastener and workpiece. By using a thread insert engineers can mix up the materials being joined, or they can choose inserts with anti-galling coatings applied.

Yet another application we see for inserts is for thread locking. There are various available styles of thread locking inserts, such as adhesive locking compounds, intentionally deformed inserts, or even proprietary styles like Spiralock (See Page 5).

Thread Insert Options

The preeminent style of thread insert are wire wound screw thread inserts. There are several brands available and several material options. They all work similarly, where you machine an oversized thread of the same pitch as your desired fastener. Then you install a pre-formed wire insert into that thread to shrink it down to the desired size. These can be very economical and effective, but some sizes are expensive and some installation tools can be extremely expensive. Screw Thread Inserts "STI" can be replaced most times if they wear out.

The next most common style are solid thread inserts. This style is essentially a threaded bushing. Where you tap a larger "standard" thread in the position you want to install the insert. Then you screw in the insert, and typically "pin" it in place to prevent it from unscrewing. These require no special machining tools, and only a very economical pin installation tool. But they do take up much more space and can be a nuisance for cleaning. They can be replaced if they wear out.

Examples of Keensert solid thread inserts.

Source: [Clarendon Specialty Fasteners]

Higbee Threads—Blunt Start

A common issue when producing threaded parts is the "knife edge" or "hair burr" formed where the end chamfer on the part and the first thread profile begin. This thin web burr can create problems when screwing parts together for the first time.

Besides careful deburring there is another solution to the problem. Adding a Blunt Start to the thread is a process of physically machining away the first thread profile until it becomes a full profile form. This almost eliminates the risk of cross-threading.

While a blunt start sounds like an obvious solution to the problem, it involves some fine adjustments and is rarely implemented.

The Higbee thread is a more formalized form of Blunt Start. In this standard, both the male and female threads require blunt starts on the "second" full form, and have the addition of a "Higbee Indicator" mark added to each part to show the Higbee Location. These are commonly used on fire hose couplings.

"This almost eliminates the risk of cross-threading"

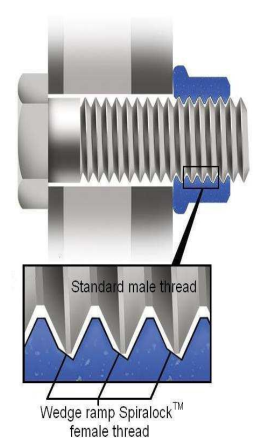
Spiralock®

Spiralock is a proprietary thread standard offered by Stanley. It was invented for application in high vibration environments, and one of the early adopters was NASA for the Space Shuttle program.

Essentially, the thread form is a distorted internal thread "Vee" which wedges the fastener in a manner to spread clamping load more evenly and resist vibrational loosening. It mates with standard bolts, and is a reuseable connection requiring no adhesives or special hardware.

With that said, Spiralock taps and thread mills are very expensive and may have low availability. We may also have to purchase special thread gages to verify the hole meets specification.

We recommend considering Spiralock Screw Thread Inserts as a more economical solution to cut Spiralock threads.



Source: [Stanley Engineered Fastening]

23rd Century CNC

9083 State Route 7
Kinsman, Ohio 44428

330-282-8111
Jon@23rdCenturyCNC.com

Find us online!

www.23rdCenturyCNC.com

What “Progressive Manufacturing” means to us...

Progressive Manufacturing is our business’s tagline and motto because it reflects our mindset.

We do not possess the most advanced equipment or tools. We do not use the most expensive software. But we do strive to make continual improvements in our processes and our people.

For example :

- *We continually invest in new inspection equipment*
- *When we add equipment, we look to add new hardware and software capabilities rather than just to replace existing ones*
- *Employees are given continual access to training programs, and are cross trained on equipment*
- *New tools are always being trialed to improve process reliability and part quality*

Please do reach out with any questions or comments regarding this article, we hope you found it helpful and entertaining!

Coating Parts with Threads

A common issue which arises on parts with threads, is how will a certain coating affect the thread.

Aluminum anodizing is a common example. Where depending on the type, this coating may add as much as 0.002-0.003” to a surface. Meaning the thread “Vee” profile may shrink as much as 0.006”!

The same goes for nearly any coating, such as nickel plating, or even certain heat treatments such as carburizing. And we have a few remedies.

The easiest in many cases is “Masking”. Where masking is simply plugging a hole, painting over, or taping off a surface to keep the coating off. But this adds cost, and negates the advantage of whatever the coating was on that area.

An aluminum part ready for anodizing with masking and plugged holes.
Source: AeroDynamics, Inc.

Alternatively, we can tap holes to larger pitch diameters so the coating builds up the surface back to nominal size. We can do this by up to +0.005” typically.

Some coatings affect thread strength negatively. Where the solution may be to first perform a Chemical Conversion coating, followed by masking, then the final coating. Hard Anodizing is one example of where this is common. Where we may first apply “Chem Film” then plug tapped holes, & finally Hard Anodize.

CENTURY CNC
PROGRESSIVE MANUFACTURING