An Easy way to Single Point Turn a Thread, version 5

By Frank Petrin as told to R. G. Sparber

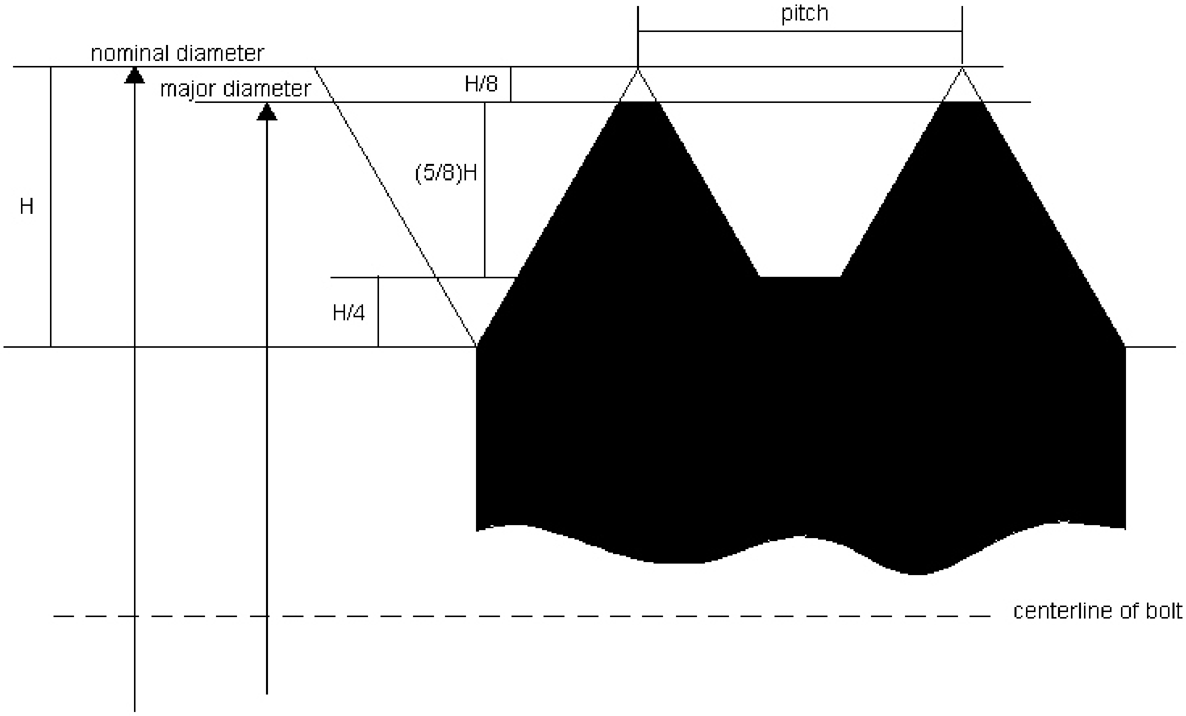
Introduction

Single point turning of a thread can seem extremely difficult until you successfully do it. One area of frustration is knowing how deep to go with the cutter. The procedure that Frank shared with me removes much of this frustration.

This is a quick and easy way to single point turn threads. It is not a precision method. If precision is needed, you will have to get a means of precisely measuring the thread as you are cutting it.

Theory

Before we cut any threads, let's look at its design.



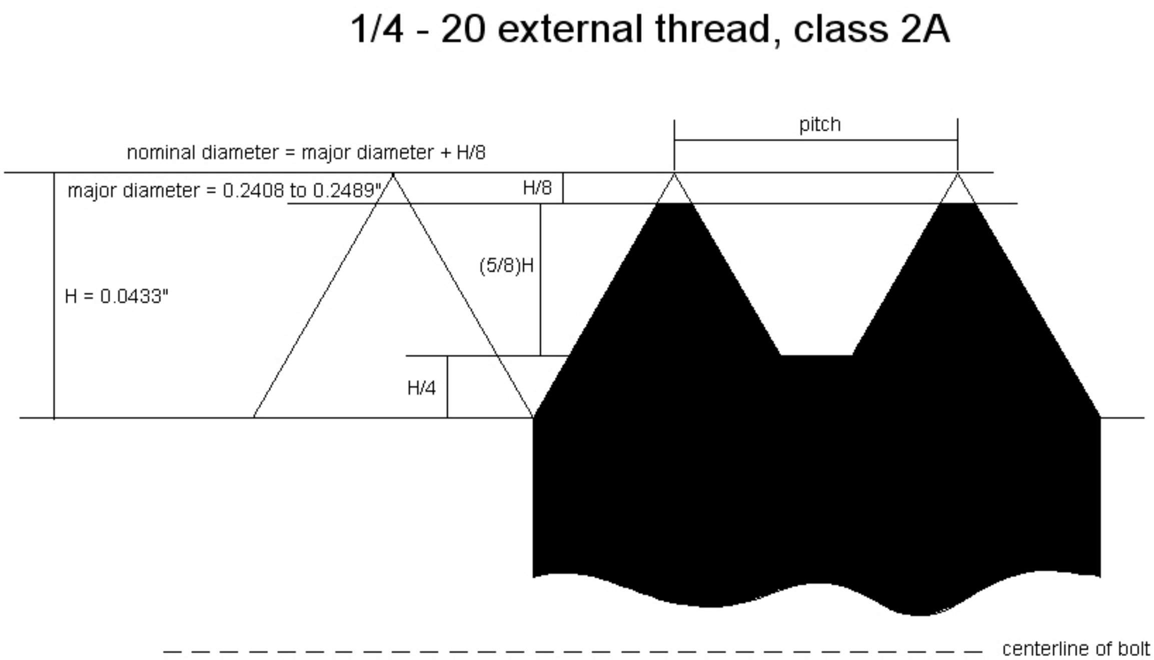
The center of the bolt is shown as that dashed line at the bottom. Not drawn to scale, we have the profile of the thread. The outer most surface of the thread is defined as the major diameter.

The pitch of a thread is the distance between identical features. I have chosen the points in this diagram. The pitch equals the reciprocal of the threads per inch. For example, if I have 20 threads per inch, then the pitch equals 1/20 = 0.05".

By design, the thread height, H, equals approximately 0.866 times the pitch. Using our pitch of 0.05", we have an H = 0.0433". This value is useful in determining the nominal diameter which equals the major diameter plus H/8. You will soon see that Frank's procedure is based on the relationship between nominal diameter and how the thread is turned.

Application

Consider a ¼ - 20 thread, class 2A. The class indicates how close a fit the external and internal parts will fit together. Class 2A is the middle one.



Since this is 20 threads per inch, we know that the pitch equals 1/20 = 0.05”. We also know that H = 0.866 times the pitch so = 0.866 x 0.05” = 0.0433”. From standard tables like those found in the Machinery’s Handbook®, we know that a 1/4 – 20 class 2A thread has a major diameter for an external thread of between 0.2408” and 0.2489”. H/8 = 0.0054”. This means that our nominal diameter can be between 0.2408” + 0.0054”and 0.2489” + 0.0054”In other words, between 0.2462” and 0.2543”. It is no accident that the average here is 0.250” as in the ¼ is ¼ - 20.

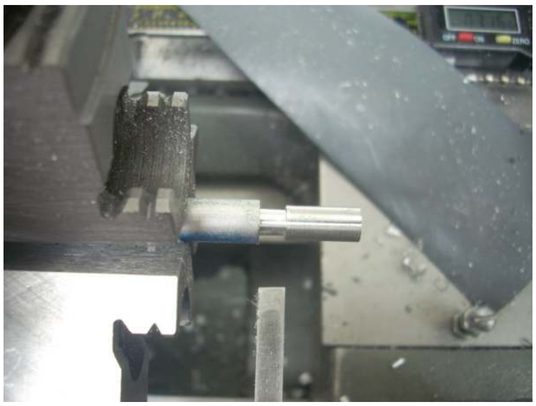
What all of this math is telling us is that if you start out with bar stock with a diameter of 0.250” and single point turn this thread until the points or edge of the thread is barely untouched by the cutter, then you have cut down the exact correct depth.

So how do we know when to stop cutting? This is where Dykem comes in.

Shop Work

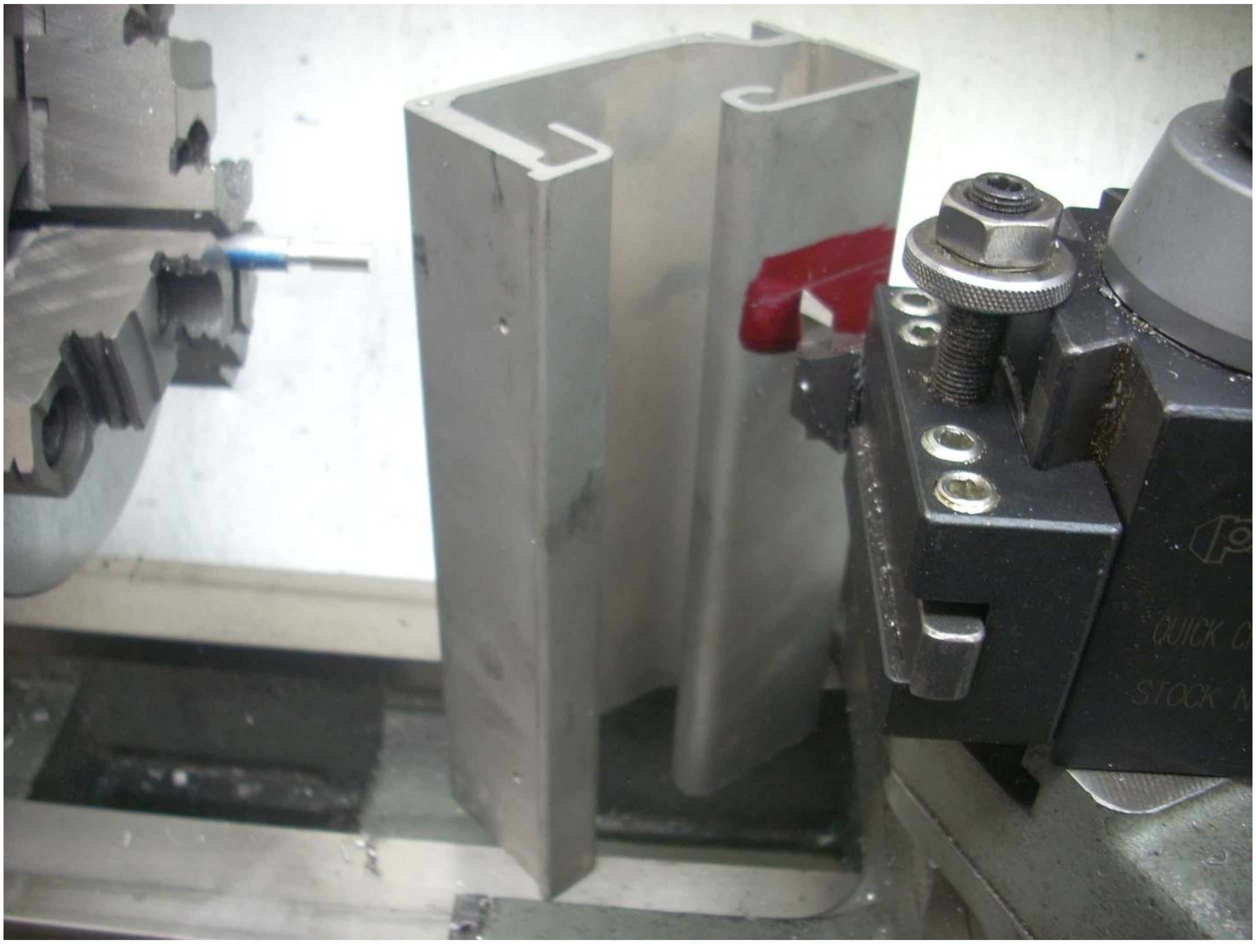
Since this is just a demonstration, I have chosen to use aluminum for my stock. Brass or steel is a more common material.

I will show you how to cut a ¼ - 20 thread using Frank's technique.

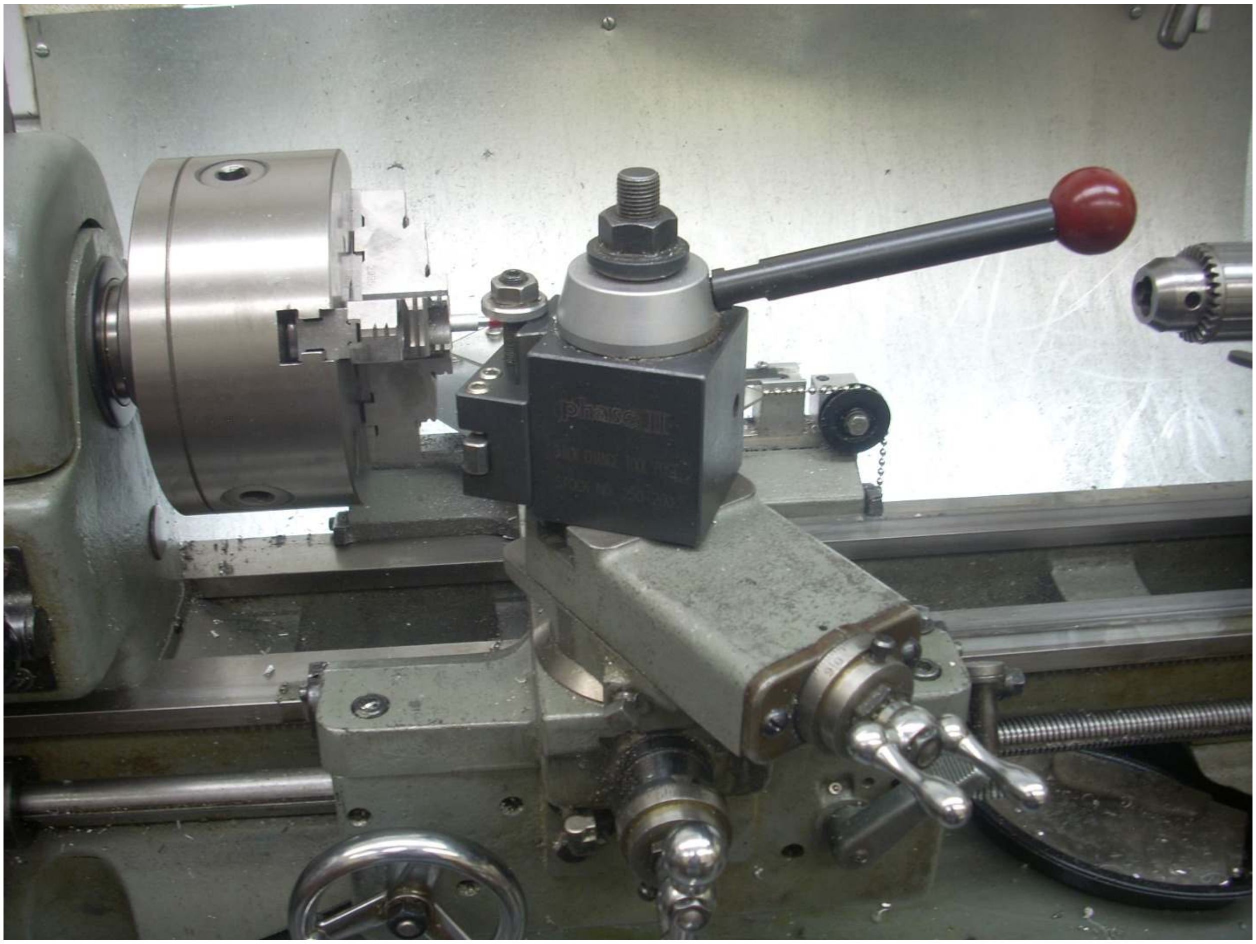


My stock is about 0.3" in diameter so I first turn it down to the nominal diameter of 0.250". The spec says it can be between 0.246" up to 0.254" for a class 2A thread.

I have also used my parting tool to cut a space for my threading tool to go at the end of the thread. This run out diameter should be no larger than the nominal diameter minus H. If we hit our nominal diameter, then the run out diameter equals no more than 0.250" – 0.043" = 0.207". I cut mine around 0.19" just to be safe. Without this run out diameter, the tool will get caught in the stock at the end of the threading pass. That can break off the tip of the cutter and/or damage the part.



I own a Diamond® tool holder which comes with a threading cutter. I use my fancy height gage to set the cutter's height to the lathe's center of rotation.



Next I set my compound to 29.5 degrees. This provides 0.5 degrees of clearance for my cutter which has the necessary 60 degree included angle point.

I am using a “fish” to align my cutter to the center of rotation of the lathe.

Larry of gingery_machines wrote to tell me -

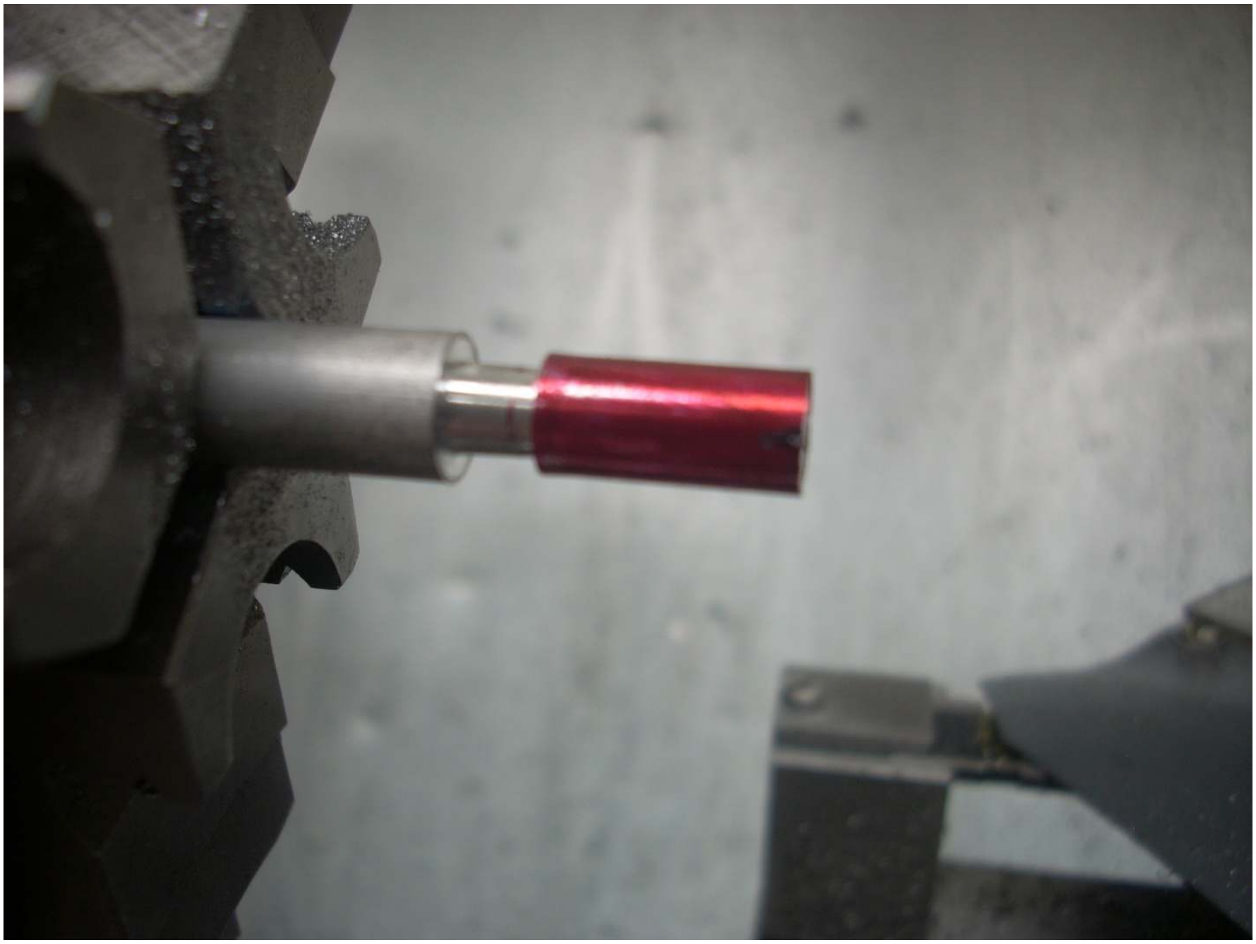
“I use the "fish", as you call it, or "fishtail" as we called it, or "center gage # C391" as Starrett calls it. Just semantics, we all know what it is. This tool is a great asset.

Not only does it help to set the tool perfect to the work as you pointed out, it also has a chart on the backside that shows the double depth of threads. This is a guide to the basic depth to cut the thread when threading straight in on the cross slide. For instance, the double depth of 12 threads is .108", so half of that is .054" which is the theoretical finish depth to cut.

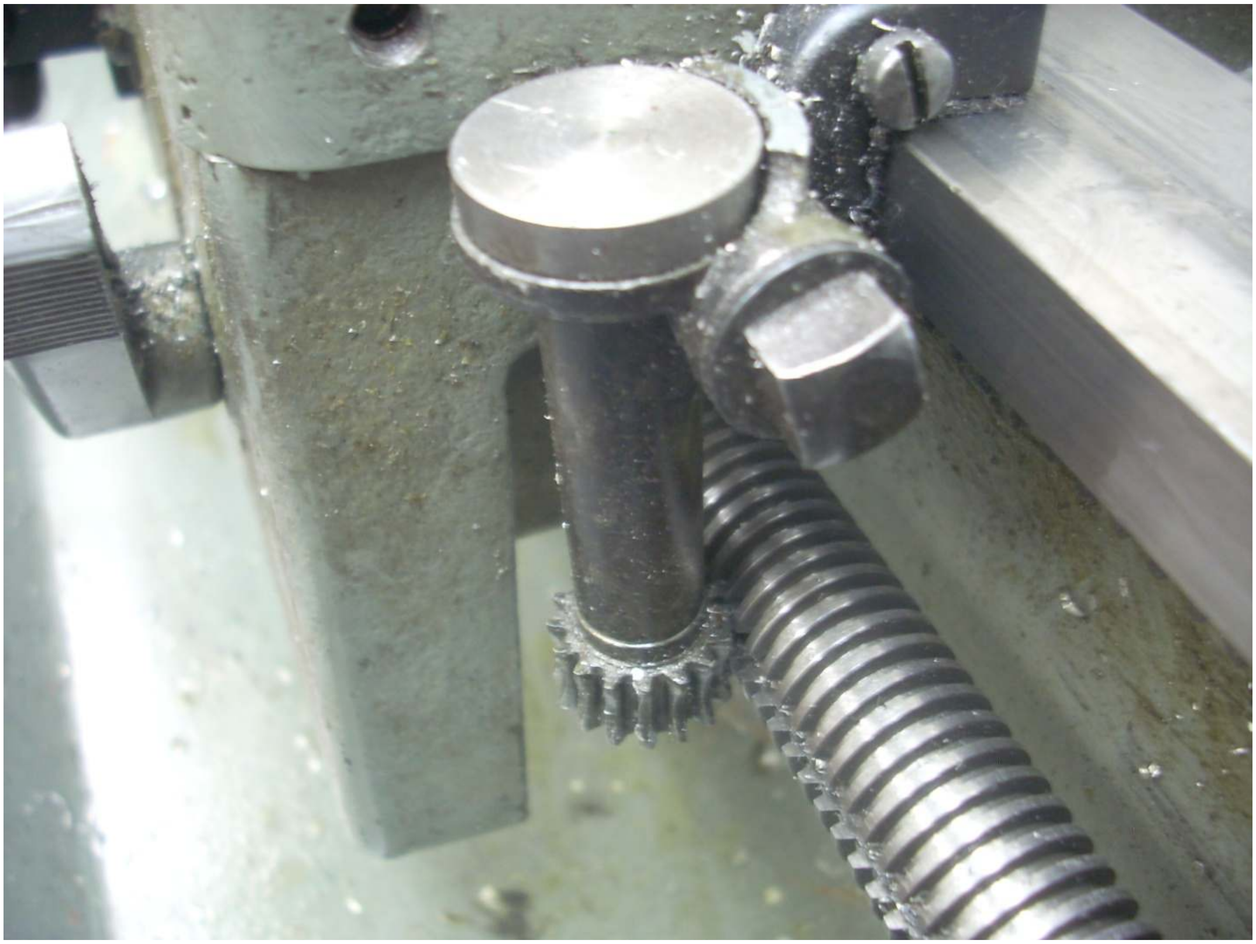
I say theoretical because I always start threading by turning the material to .005" under the nominal size for work from 1/4" to 3/4" diameter, a bit more for larger work. I have a gage I made to cut 2 1/8" - 5 TPI. That thread is cut .010" undersize on the major diameter.

The actual finish size for the thread will be when-
A) It fits the mating part, or
B) It measures correctly using thread wires.

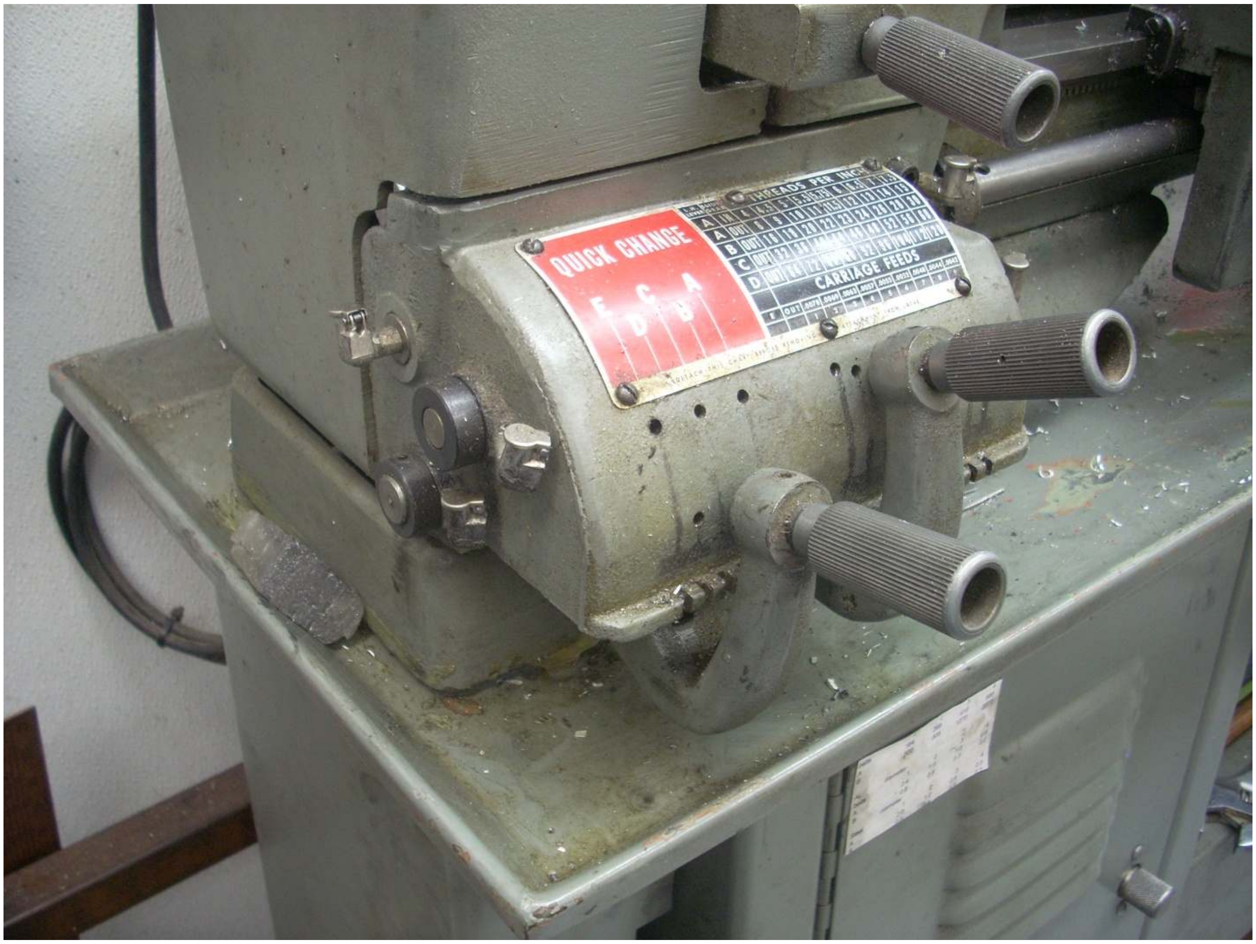
One more thing about the "fish"- it has scales (pun intended) on its edges in increments of 32, 24, 20, & 14. This will help verify TPI in the absence of a proper thread gage.”



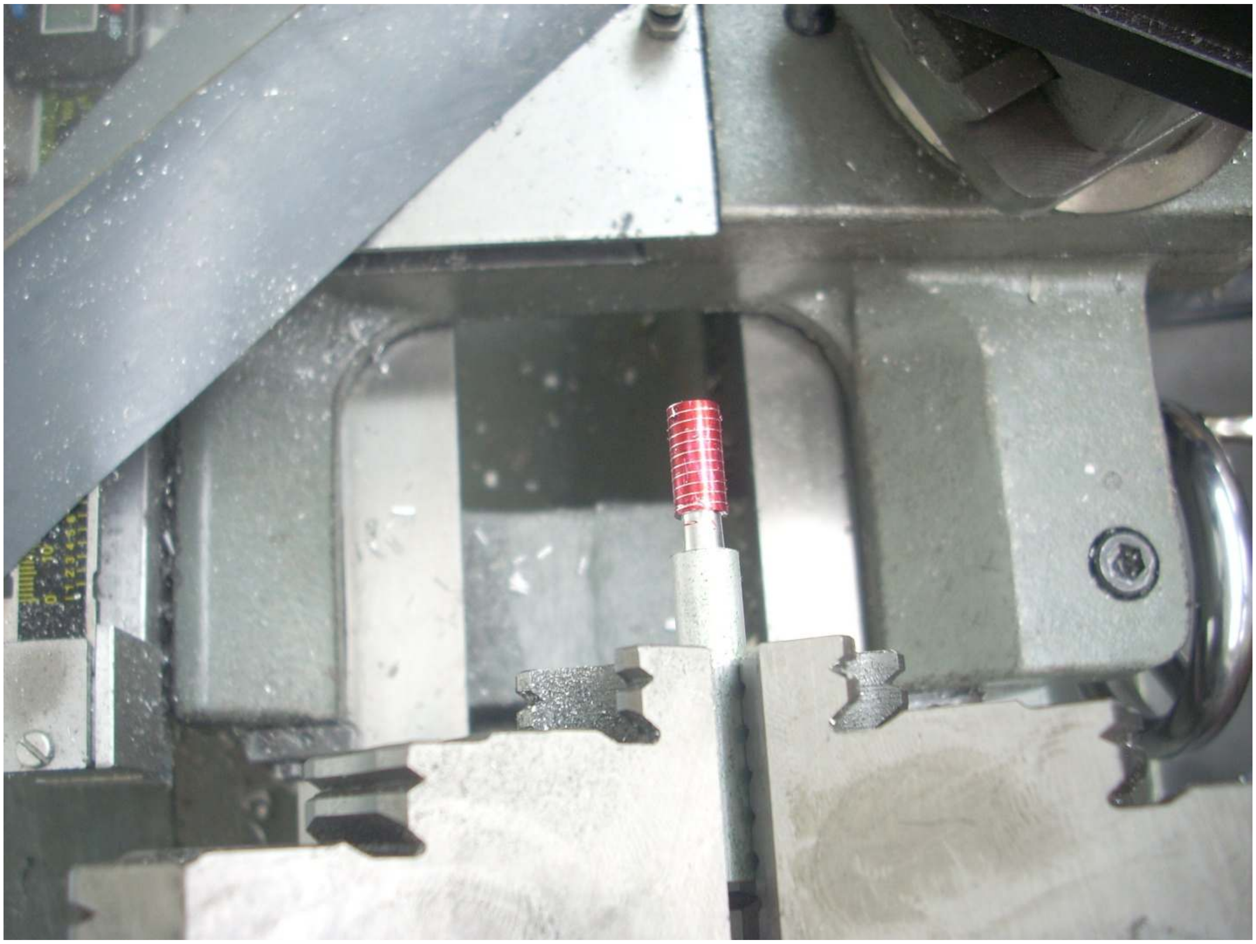
Now the all critical step – coat the surface to be threaded with Dykem.



I am now ready to set up the lathe for threading. The “clock” is engaged. It will tell us when to lock in the split nut.

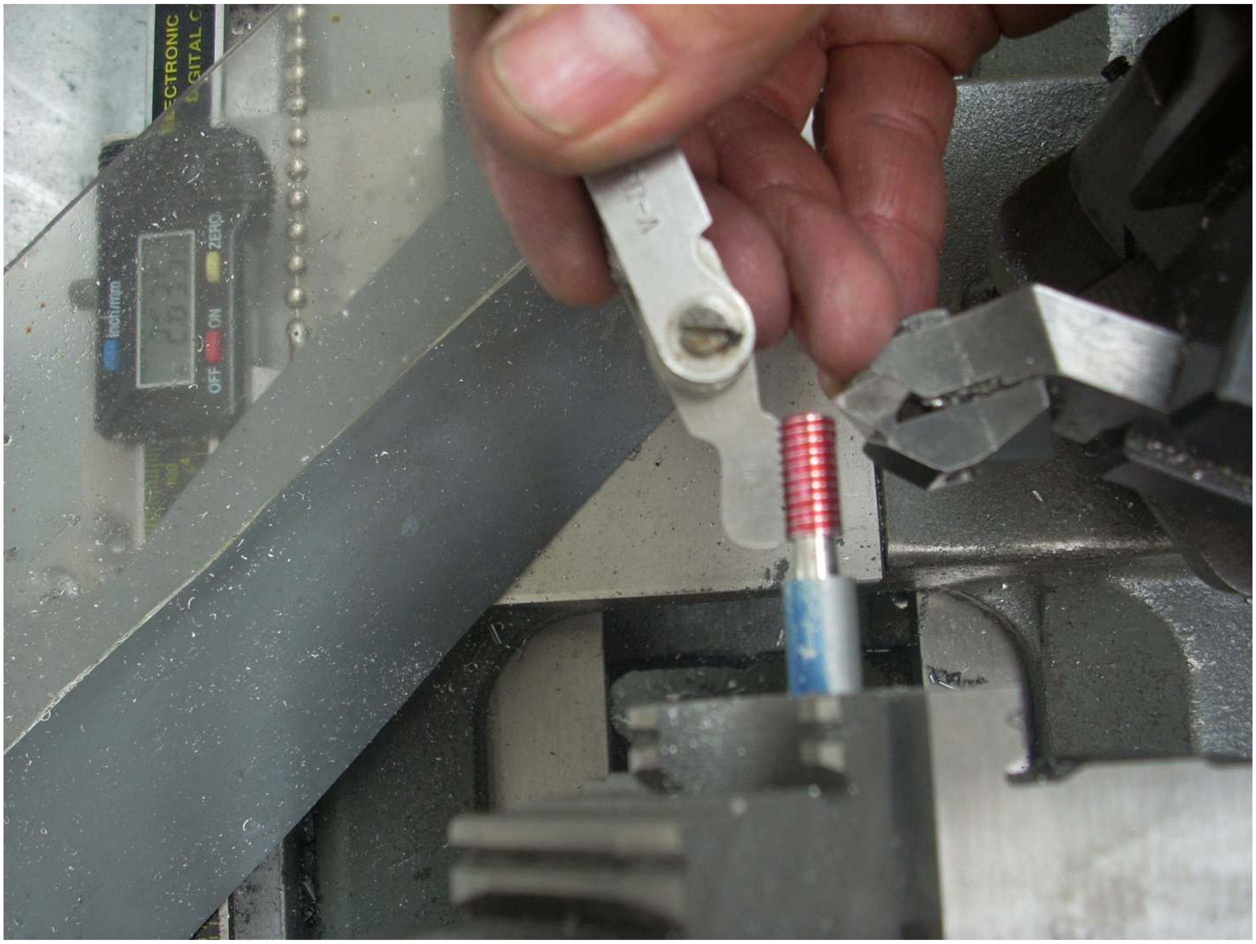


I set my Quick Change Gear Box for 20 threads per inch.
My lathe can go as slow as 26 RPMs by using back gears and that is where I set it.
With practice, you can go faster as long as you can disengage the split nut before the cutter leaves the run out area.

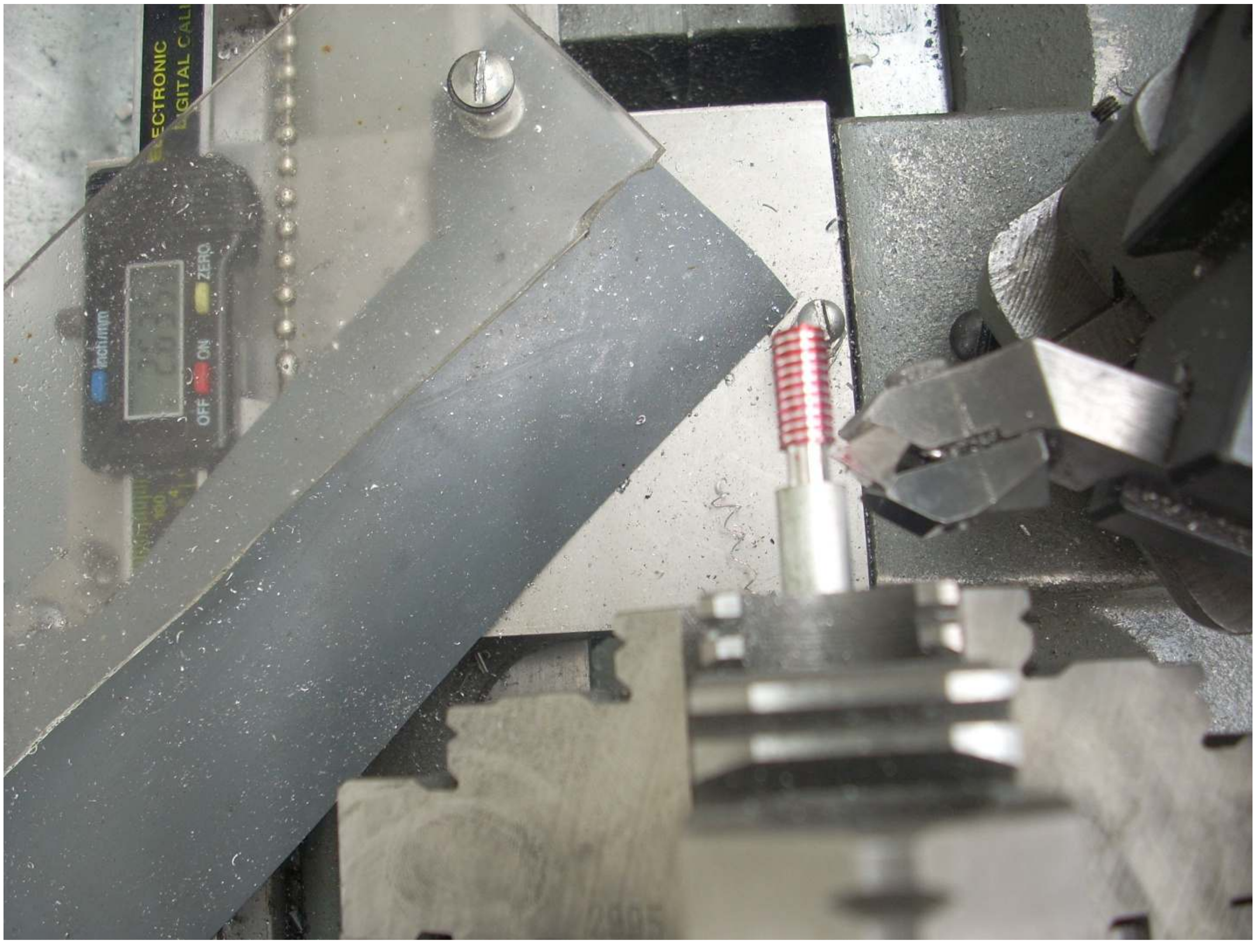


I move the cutter so it just touches the part and zero both my cross slide and compound dials. Then I back out the compound and move the apron over so the cutter is to the right of the part. And finally, I turn the compound's dial back to zero.

Next I make a very light cut with both dials at zero. I start the machine and wait for one of the clock's tick marks to line up with the arrow. Then I engage my split nut. You can barely see the shiny aluminum through the Dykem.



Using my thread gage, I verify that I am cutting 20 threads per inch.



I feed in 0.005" on my compound for the first pass.

I follow with three more passes at 0.005". The cutter is starting to strain the part a little so it is time to feed in a little less. I take two passes at 0.004" before changing to two passes at 0.003". The rest of my passes are at 0.002". My lathe is not ridged enough to take 0.001" deep cuts reliably. The cutter often jumps out and just rubs the surface.

After each pass I can see the Dykem line getting thinner.

After six passes at 0.002" each, the Dykem is essentially gone.

I have changed out my cutter for a standard one and will now reduce the diameter to the major diameter of 0.2408" to 0.2489". My goal is to use a 1/4 - 20 nut I have in hand so ultimately, it must fit.

Reducing the diameter of the threads will cause a burr to be generated. It will be pushed down into the thread and must be removed. A thread file would be useful here[2].

I start by turning down to 0.248" but the nut won't fit. So I turn down to 0.241" and it goes on about ¼" but then jams. In the end I turned it down to 0.237" and the nut easily spun on.

So what went wrong? Referring back to the standard tables I see that 0.237" is the minimum major diameter for a class 1A thread. So nothing was really wrong here, I just did not realize that my nut was a class 1A and not 2A.

[2] For example, http://www.use-enco.com/CGI/INPDFF?PMPAGE=123&PARTPG=INSRIT

There is one more step that nicely finishes the job. I grind or cut a 45 degree taper on the end. This makes threading on a nut much easier and gives it a professional look.

Acknowledgements

Thanks to Frank Petrin for generously sharing this technique with me. Thanks to Larry of gingery_machines for the lesson on how to use the “fish”.

I welcome your comments and questions.

Rick Sparber
Rgsparber@aol.com

<u>Frank’s Single Point Threading</u>

1. Define terms:
    a. Nominal diameter N
    b. Threads Per Inch TPI (N-TPI as in ¼-20)
2. start with nominal diameter
3. cut run-out area to a depth of $\frac{0.866}{TPI}$ inches
4. coat area to be threaded with dye
5. set compound to between 27° and 29.5°
6. set threading cutter true to part using “fish”
7. back out compound; feed in to remove backlash; set dial to 0
8. feed in cross feed until cutter touches part; 0 dial
9. set lathe to cut TPI
10. cut thread
    a. feed in 0.005” on cross slide
    b. make pass
    c. repeat until cutter loads up and then reduce the feed by 0.001”
    d. stop when dye no longer visible
11. reduce the diameter of the part by $\frac{0.217}{TPI}$ $inches$
12. deburr thread

R.G.Sparber August 22, 2011