

A Close Look at Single Point Threading, version 8

By R. G. Sparber

Copyright protects this document.¹

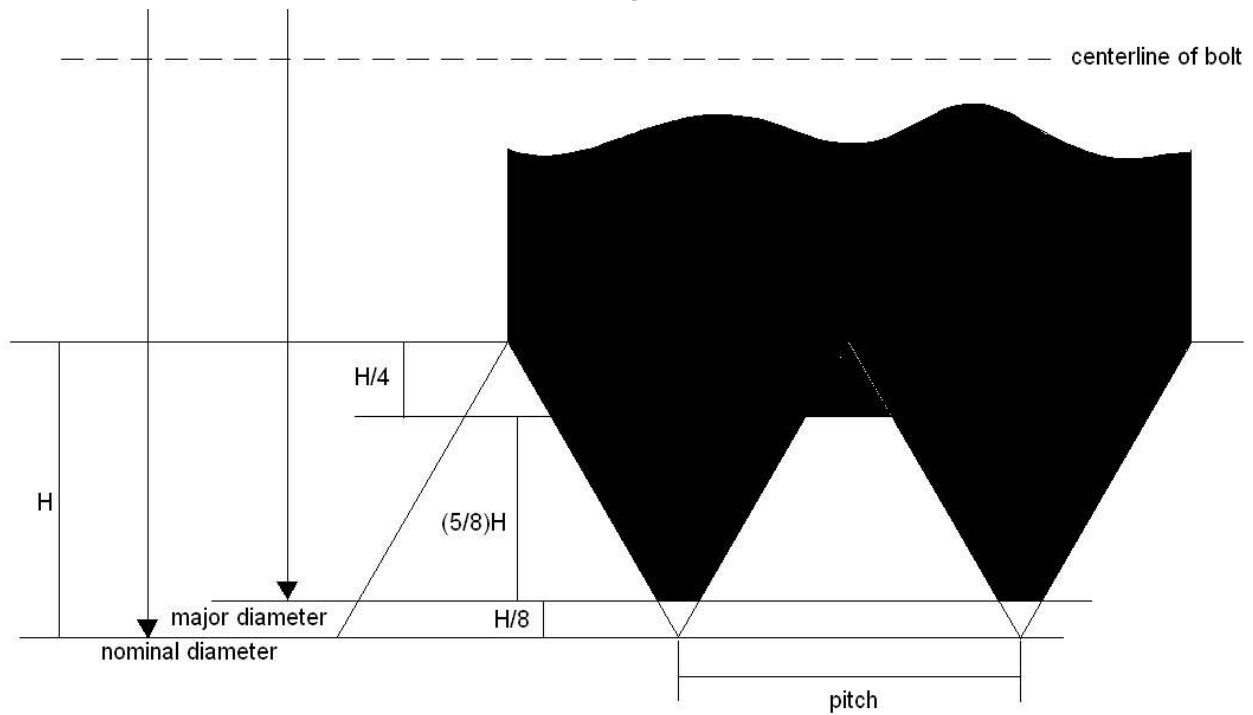


Contents

A review of the Theory	2
The Cutter.....	3
The Parting Tool Method.....	5
The Traditional Method	8
A Compound Set Over Greater Than 30°	13
A Compound Set Over Less Than 30°.....	15
The Standard Case.....	17
The Non-standard Case (just for doc).....	18
The Malcolm Method.....	19
That Professional Look and Feel.....	23
Bench Work.....	23
Reminder Card	28
Acknowledgements	30

¹ You are free to copy and distribute this document but not change it.

A review of the Theory



The center of the bolt is shown as that dashed line at the top. Not drawn to scale, we have the profile of the thread. The outer most surface of the thread is defined as the major diameter. This is the view as you stand in front of the lathe and look down at the thread in front of the cutter.

The pitch of a thread is the distance between identical features. I have chosen the points or crest 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 $\frac{1}{20TPI} = 0.05''$.

By design, the thread height, H, equals approximately 0.866 times the pitch.

No matter how you run the cutter, the end result must look like this for an external thread². The thread profile is essentially a V groove with the top and bottom missing. If it helps your understanding, unwrap the thread from the bolt and see it as a long groove. The cutter runs along this groove. This mental model does not explain the pitch but should be otherwise correct.

² This is an American National Standard Unified External Screw Thread.

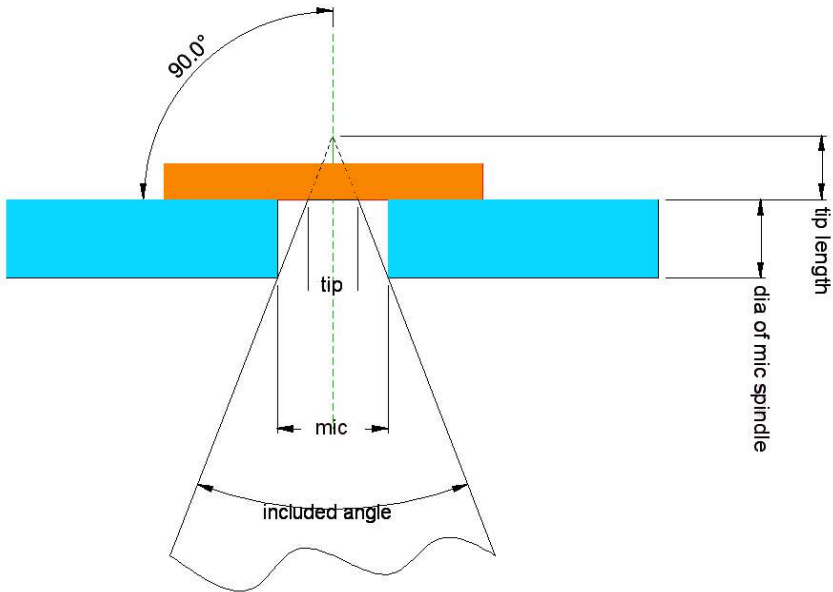
Regardless of method, the cutter's main axis must be perpendicular to the longitudinal axis of the lathe. This axis is the same as the center of rotation.

The Cutter

I am using a threading tool that has a 60° included angle. This shape enables me to cut an American National Standard Unified External Screw Thread profile. Any thread can be cut if you grind the correct profile into a cutter. This is where High Speed Steel cutters really shine. They are easy to grind to any profile.

One challenge is to form the tip of the cutter accurately. Malcolm of [gingery_machines](http://www.youtube.com/watch?v=ma6kdIHwrFI) provided me with an answer. His YouTube video (<http://www.youtube.com/watch?v=ma6kdIHwrFI>) is very helpful but I have to admit, I could not follow the math. So here is my try at it.

One note about this procedure – it does not work on cutters made from $\frac{1}{4} \times \frac{1}{4}$ tool blanks given an anvil diameter of 0.25". There isn't enough angled flank on the cutter.



The cutter has been ground with a known included angle. The tip is on the center line of the cutter. The diameter of the mic's spindle is known. The cutter is held so its centerline is perpendicular to the mic's spindle. A piece of flat stock is resting on the back side of the spindle and the flattened tip of the cutter. The

mic has been adjusted so the anvil and spindle just contact the cutter.

$$tip\ length = \frac{mic}{2 \tan\left(\frac{included\ angle}{2}\right)} - diameter\ of\ mic\ spindle$$

$$tip = 2 \times tip\ length \times \tan\left(\frac{included\ angle}{2}\right)$$

If the included angle = 60° and the diameter of the mic spindle is 0.25", then

$$tip\ length = \frac{mic}{1.155} - 0.25"$$

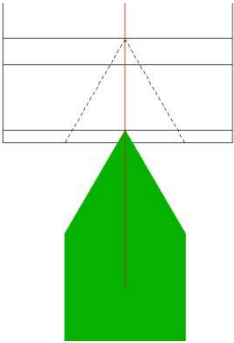
$$tip = 1.1547 \times tip\ length$$

Since $tip\ length = \frac{H}{4} = \frac{0.866P}{4}$, and $pitch = \frac{1}{TPI}$ we can also write the mic reading for a given Threads Per Inch:

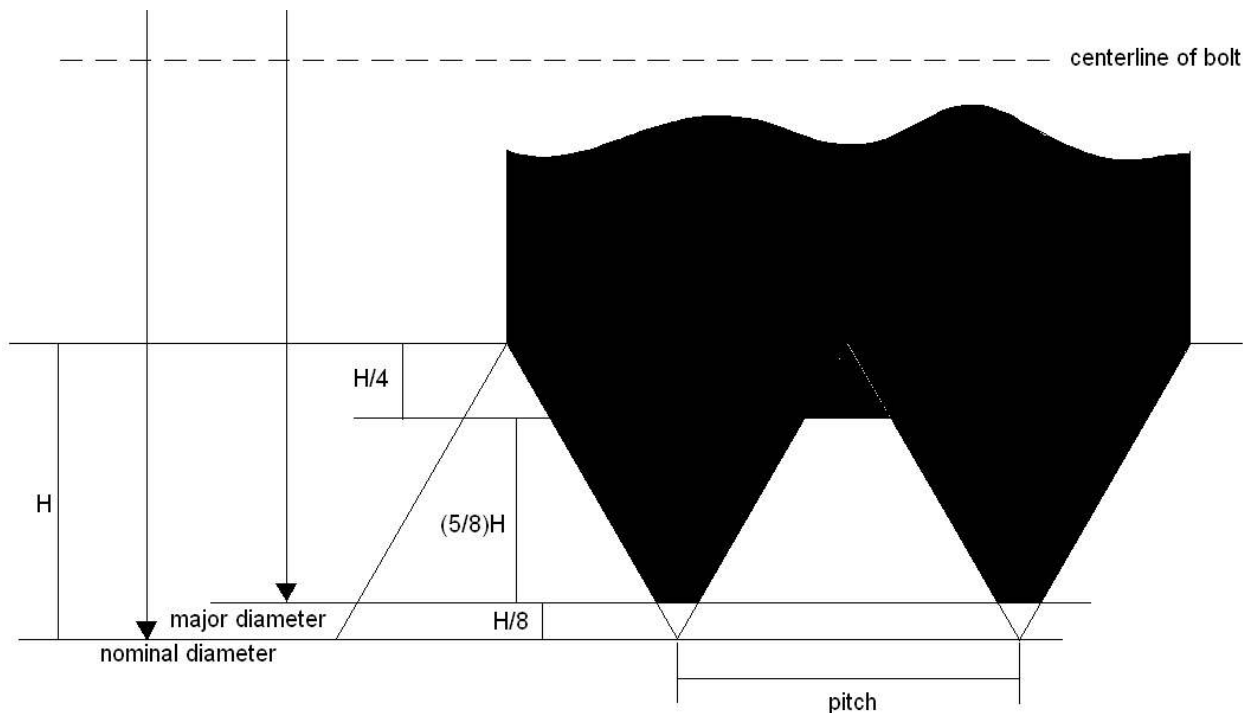
$$mic = \frac{1}{(4\ TPI)} + 0.289"$$

The tip should equal $\frac{1}{4\ TPI}$.

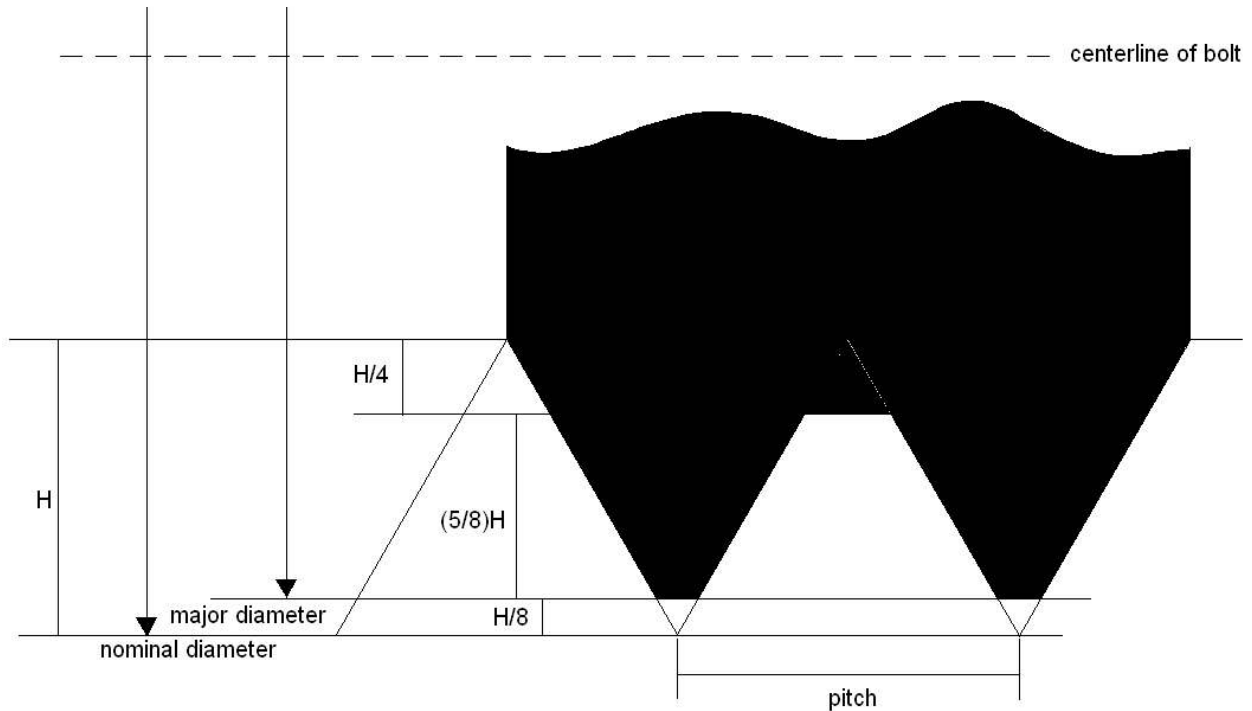
The Parting Tool Method



One way to feed in the cutter is like a parting tool. The cutter is advanced a small distance and the first pass is made across the cylinder. Then the cutter is backed out, moved to beyond the right end of the part, and the cutter turned in a small distance beyond the first pass. This process is repeated until the proper depth has been achieved.



Recall that $H = 0.866Pitch = \frac{0.866}{TPI}$. Given that the tip of the cutter has been reduced by $\frac{H}{4}$ and the round stock has a diameter equal to the major diameter, then the tool must be fed in a total of $\frac{5}{8}H$.



For a $\frac{1}{4}$ - 20 thread this means that $H = \frac{0.866}{20TPI} = 0.043''$; the tip has been reduced by $\frac{0.0433}{4} = 0.011''$; and the total feed in of the tool is $\frac{5}{8} \times 0.043 = 0.027''$.

If the stock starts with a nominal diameter, then the total feed in of the tool = $\frac{1}{8}H + \frac{5}{8}H = \frac{3}{4}H$. But then when done cutting the thread, you must reduce the part by a radius of $\frac{H}{8}$ to the major diameter so the crest is correct.

Donald of the gingery_machines BBS points out: “The usual argument against this method (i.e. in favor of setting over so you're feeding effectively parallel to one face) is to avoid the cut material from both sides of the tool bit from crowding; this crowding causes an increase in pressure and can lead to chatter or digging the cut-off material into the face and pushing the work piece out of alignment, leading to a tapered thread.”

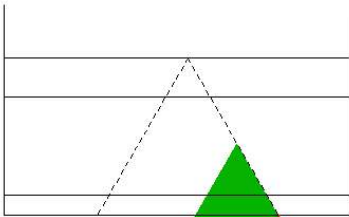
Russ of atlas_craftsman pointed out that with the cutting force on both sides of the cutter, there is no consistent pressure on the leadscrew and halfnuts. The result is that any backlash will show up as a wandering thread. The methods shown below don't have this problem.

Doc of atlas_craftsman wrote – “... I believe the helix on the lead screw provides constant pressure towards the headstock, eliminating backlash or a

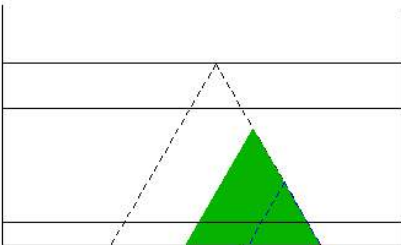
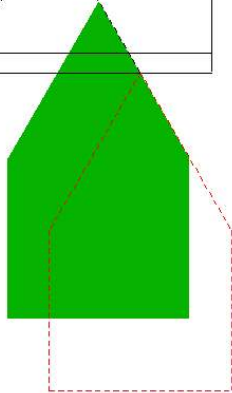
drunken thread ...the straight in bit has pressure perpendicular , but there is pressure to the left by the driving of the lead screw...” So I guess Russ and Doc will have to go outside and settle the matter ;-)

The Traditional Method

The compound is set to an angle less than 30° but typically greater than 27° . For the sake of clarity, let's say it is 30° for now. Recall that the cutter has a 60° included angle. On each pass, the compound is advanced into the work. Note that the right face slides along the cut side of the V. The left face does all of the cutting.

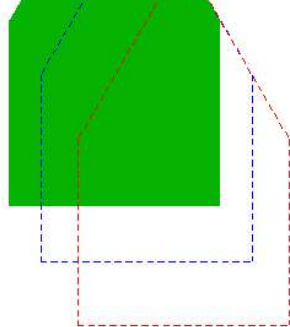


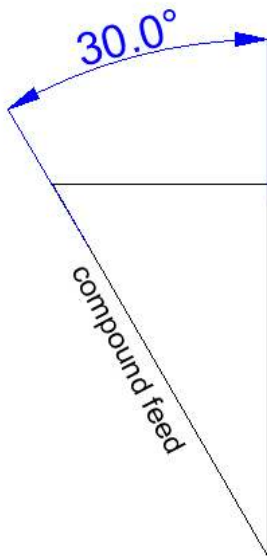
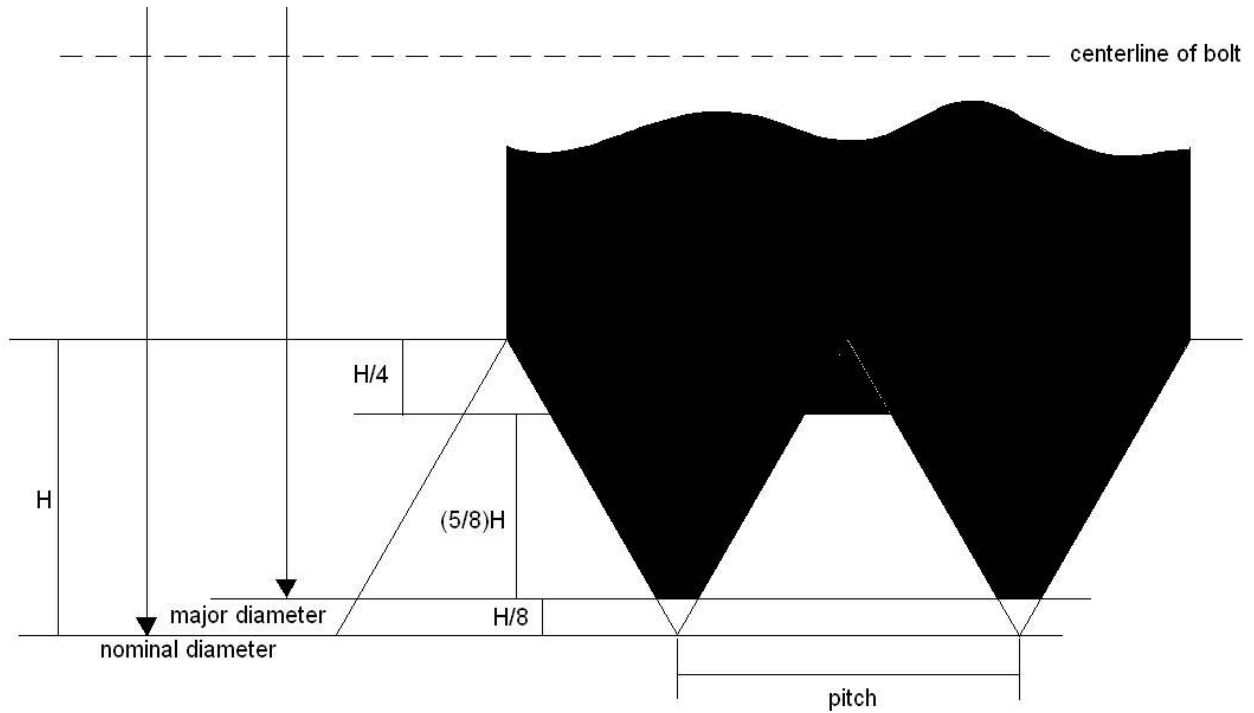
We start with the point of the cutter on the surface of the cylinder. Then, with the cutter off of the part, we feed in a small amount, typically 0.005".



The process is repeated as we march down the left side of the groove that will become our thread.

Note that in the Parting Tool Method, we are advancing the cross feed dial and in the Traditional Method it is the compound dial.

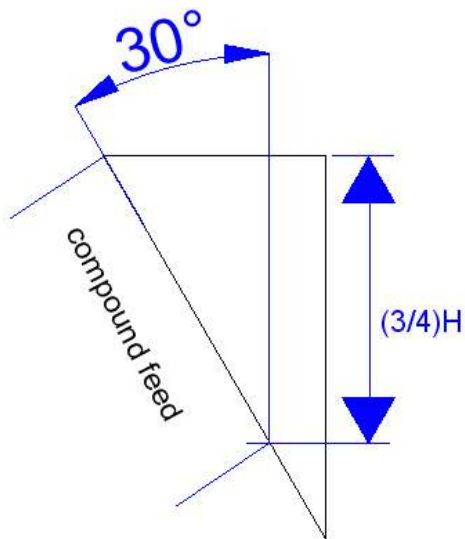
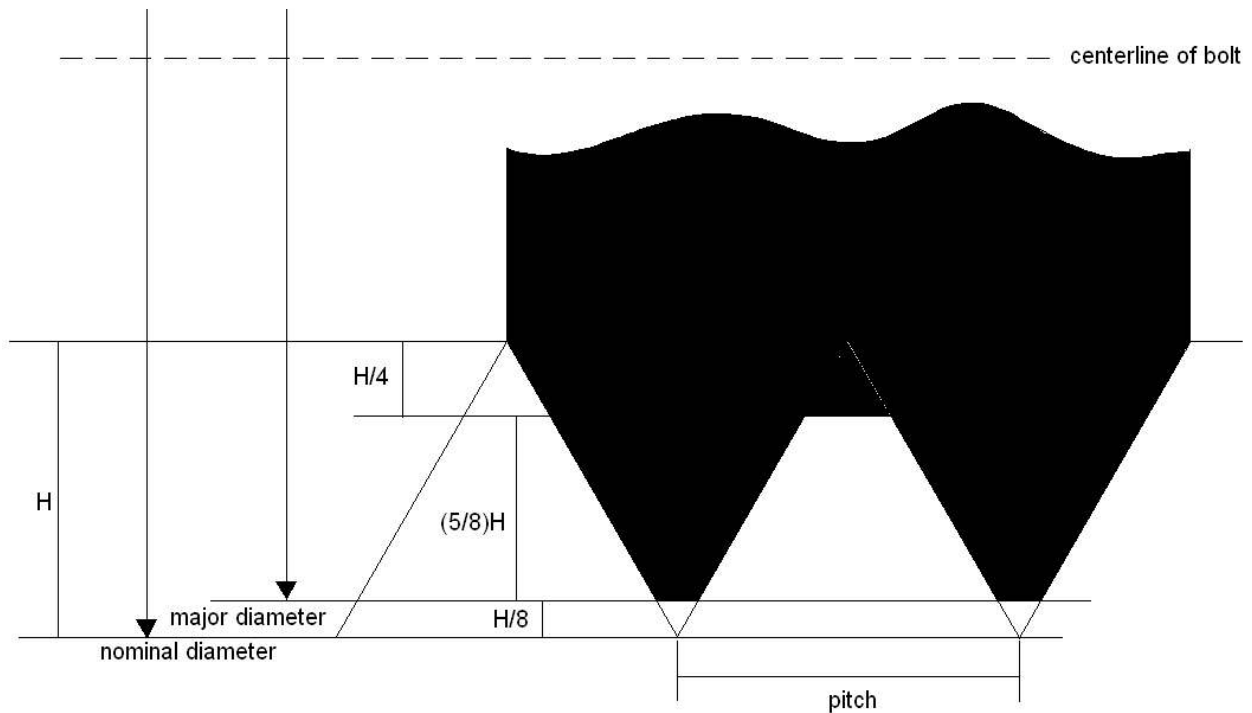




The thread depth, $H = \frac{0.866}{TPI} = 0.866 \times \text{pitch}$.

The total compound feed is $\frac{H}{\cos 30^\circ} = \frac{H}{0.866} = 1.155H = \frac{1}{TPI}$.

H But this assumes we are starting at the nominal diameter and are ignoring the shape of the root. That won't get us our standard thread cross section.



If we are starting at the nominal diameter and the cutter's tip has been shortened by $\frac{H}{4}$, then we must

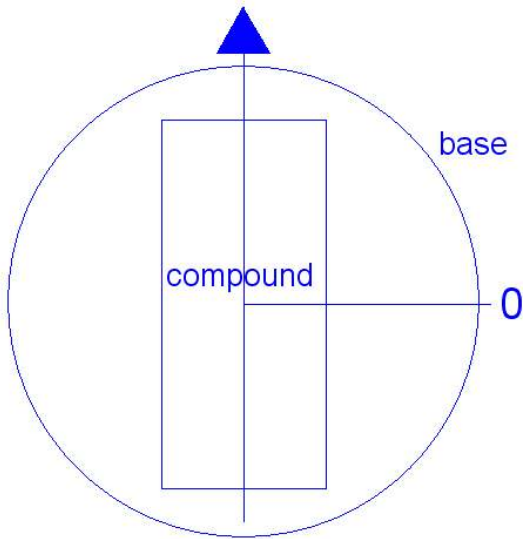
$$\text{feed in } \frac{\frac{1}{8}H + \frac{5}{8}H}{\cos 30^\circ} = \frac{3H}{4 \cos 30^\circ} = 0.866H = \frac{3}{4TPI}$$

When done with all passes of the cutter, we must remove $\frac{H}{8}$ from the outside radius. This can be done with a cutter but note that a burr may be raised. Something like a thread clearing file will be needed. An alternate method, mentioned by CT2 of gingery_machines is to put a file on the surface with the lathe running. You are not

removing much metal and you avoid raising a burr.

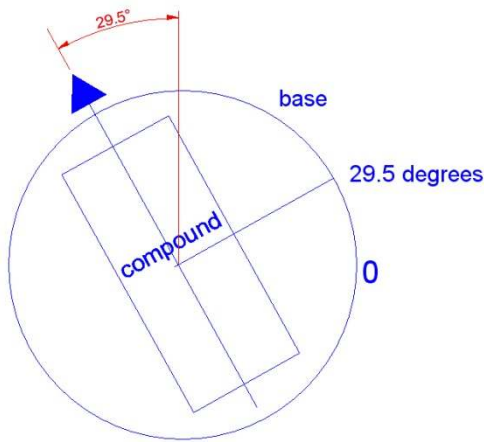
I will return to this subject later.

Next, let's look at having the compound at other than 30°. First, I better define what I mean by this angle, at least on my Atlas-Craftsman lathe.



Here is a top view of my compound. The circle represents my graduated scale which reads out in degrees. On both sides of my zero point the numbers increase. With the compound set to 0°, my axis of movement is perpendicular to the longitudinal axis of the lathe.

front of lathe

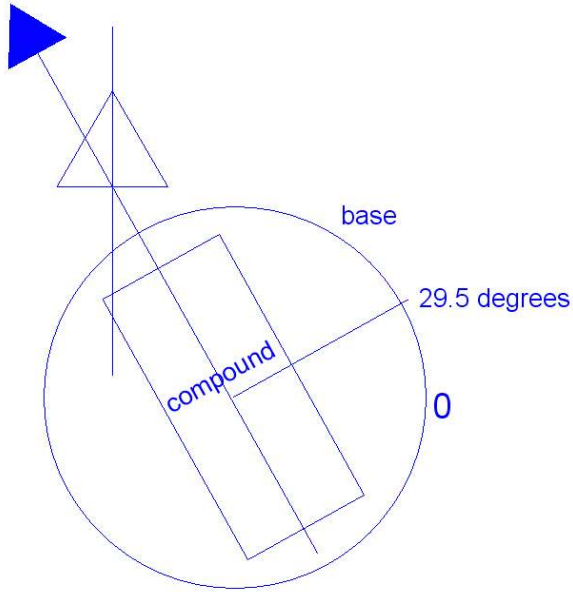


When I set over my compound to 29.5°, it looks like this. My dial and pointer which is on the side reads the angle which is the same as the angle formed by the compound's axis and a line perpendicular to the longitudinal axis.

Not all lathes define 0° as perpendicular to the longitudinal axis. Some have 0° defined as parallel to the longitudinal axis.

When in doubt, get a picture!

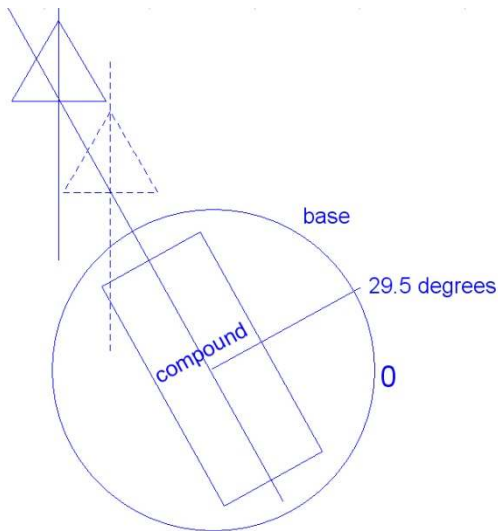
front of lathe



front of lathe

My solid head arrow shows the path of anything supported by the compound.

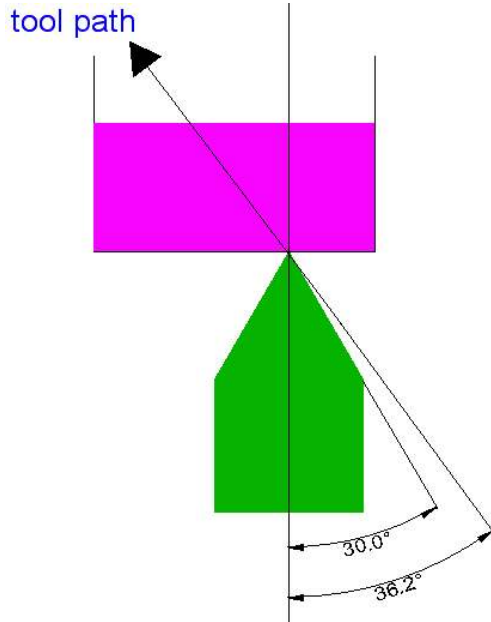
As I crank the compound's dial, my cutter, shown as a transparent triangle, moves along this path. This means that the cutter moves both in, towards the center of rotation and to the left towards the headstock.



front of lathe

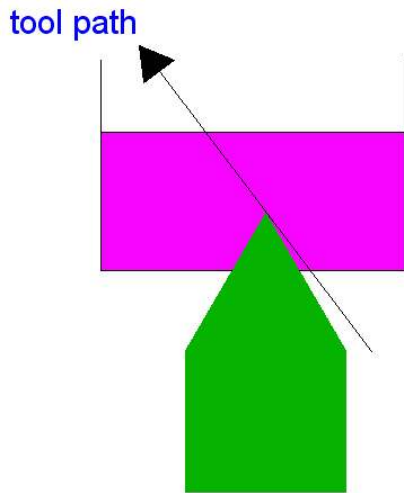
Here you see the cutter advanced by the compound. The dashed lines represent the starting point and the solid triangle is the present position. Note that the right flank of this triangle, that is, cutter, lines up. This is just another view of what was shown on page 8.

A Compound Set Over Greater Than 30°

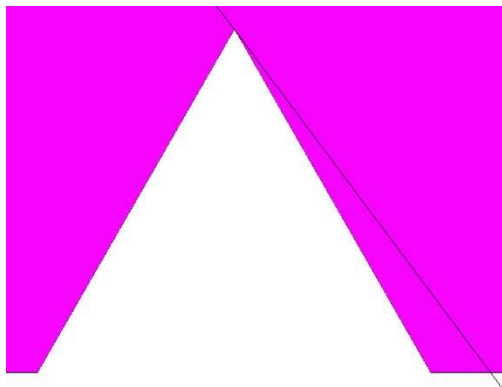


It is hard to see a $\frac{1}{2}$ degree difference so let me exaggerate the picture.

The right face of the cutter is at an angle of 30° with respect to the lathe's cross feed axis. The tool path, as set by the compound angle, is at **greater** than 30°, I chose 36.2°.

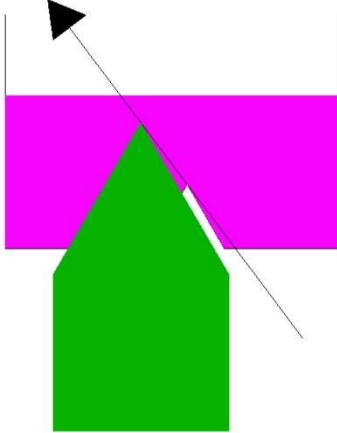


At the end of the first pass, the cutter has followed the tool path which is at an angle of 36.2°.



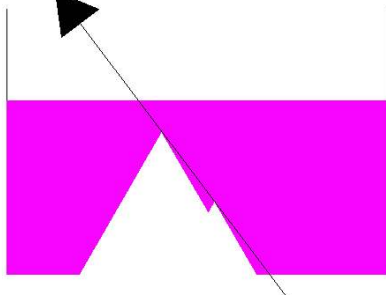
If I pull the cutter out and look at the cut, I can see that the tip is right on my tool path line which is at 36.2° but the right flank has cut at 30°. Note that the V that was cut does have the desired 60° included angle.

tool path



Let's take another pass with the cutter. When I advanced the cutter with the compound, my tip will again follow the tool path at a 36.2° angle.

tool path



Pulling the cutter out we can see how the first cut compares to the second. We get this nasty ridge between cuts and clearly this is not the uniform left flank that we need.



Here is a photograph of a thread cut with the compound angle greater than 30° . You can clearly see the ridges on the right side. The left side doesn't look so great either.

Russ, the author of the picture, wrote:

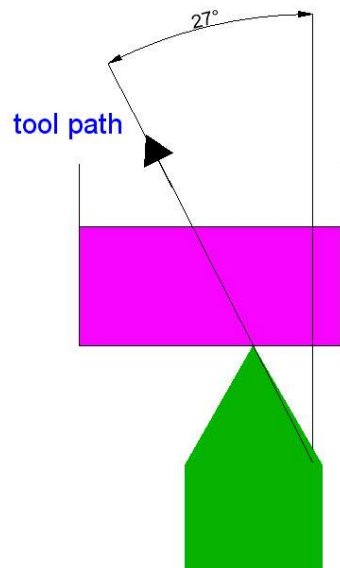
“I do have to comment on the picture of the cut thread: at the end it was taking a $.060$ DOC³ cut with $.020$ chip on a threading insert, some stripping of the base material would be expected. I don't think I've cut a thread like that in 20 years, I should have run a cleanup pass or 2.”

Picture by Russ Kepler used by permission.

³ DOC means Depth Of Cut.

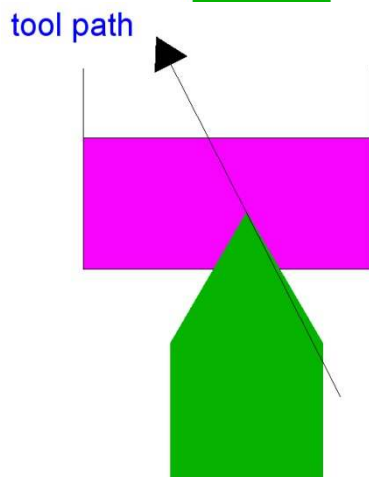
OK, so we now know how *not* to set the compound and what happens if we do.

A Compound Set Over Less Than 30°

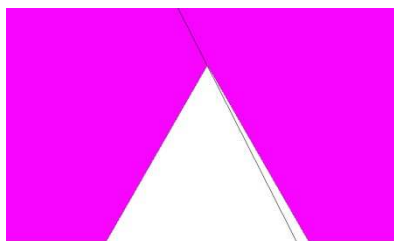


Say our compound is correctly set to *less* than 30°. Then the tool path is set to less than 30°. I have chosen 27°.

We start with the tip of the cutter on the surface of the cylinder. I have drawn the tool path line through this tip.

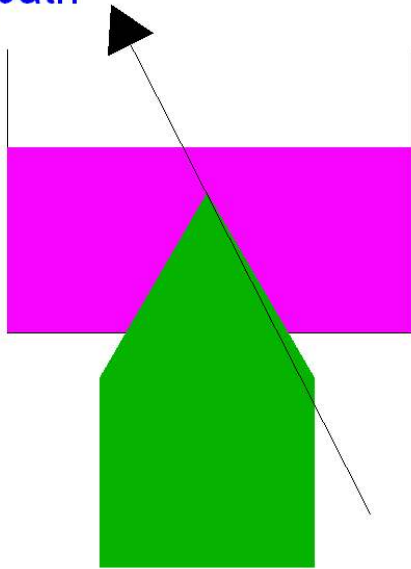


With the cutter to the right of the cylinder, the compound is turned to advance the cutter towards the center of rotation. Then the first pass is made.

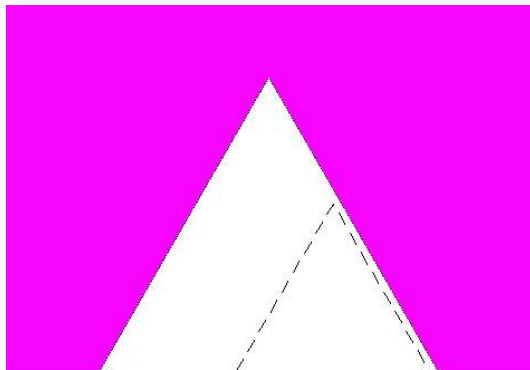


When the cutter is withdrawn, we see the V made by the cutter. The bottom of the V is aligned with the tool path line.

tool path



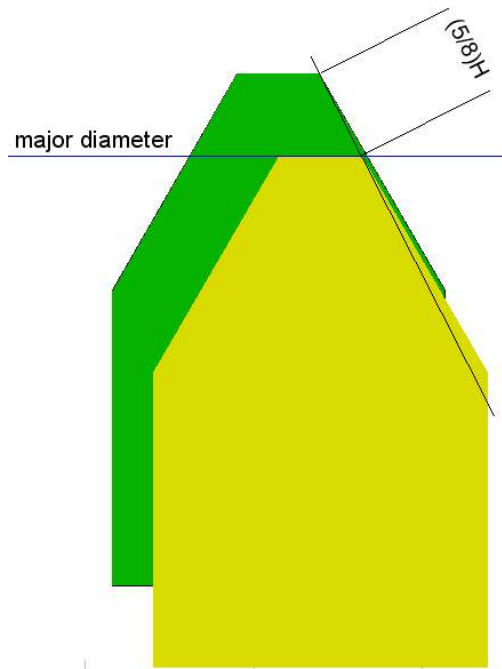
The second pass of the cutter is made. We again see that the tip of the cutter is on the tool path line.



When the cutter is backed out, we see this second cut. I have shown the first cut with a dashed line.

Note that the outline of the first cut is entirely within the second cut. This means that the second cut removed metal on both flanks of the cutter and freshly defined the V.

The Standard Case



Now we will put it all together. The compound is set over at an angle of β° where β° is less than 30° .

The tip of the cutter has $\frac{H}{4}$ removed from its length.

The cutter touches down at the major diameter.

We feed the cutter in a total of $\frac{5}{8}H$ on the radius which means the compound is turned in a total of $\frac{\frac{5}{8}H}{\cos\beta^\circ}$.

Recall that $H = \frac{0.866}{TPI}$ so we can say that the total compound infeed = $\frac{0.541}{(TPI)(\cos\beta^\circ)}$.

If the compound is set over 29° , then

$$\text{the total compound infeed} = \frac{0.541}{(TPI)(\cos 29^\circ)} = \frac{0.619}{TPI}$$

For example, if we are dealing with 20 TPI, then

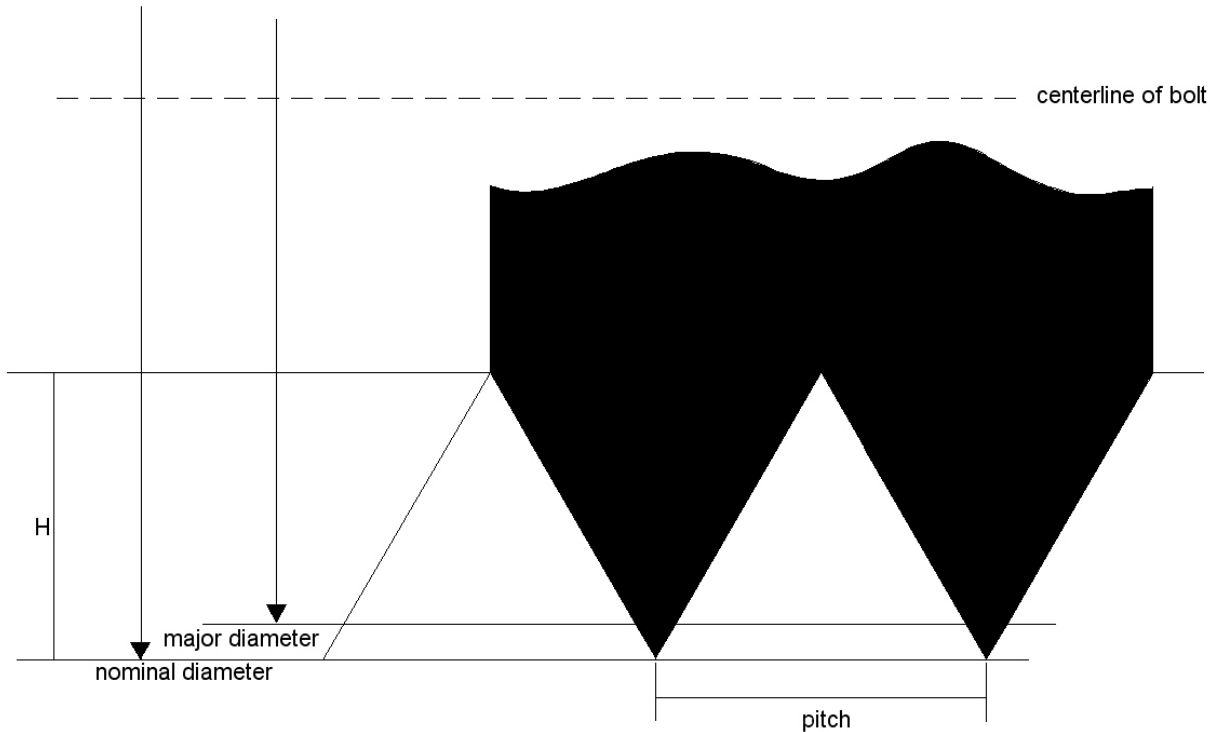
$$\text{the total compound infeed} = \frac{0.541}{(20)(\cos 29^\circ)} = \frac{0.619}{20} = 0.03093'' \approx 0.031''$$

Note that if the set over was 30° , this total compound infeed would be

$$\frac{0.541}{(20)(\cos 30^\circ)} = \frac{0.619}{20} = 0.03123'' \text{ versus } 0.03093''.$$

So for practical purposes, the total infeed is the same for 29° or 30° .

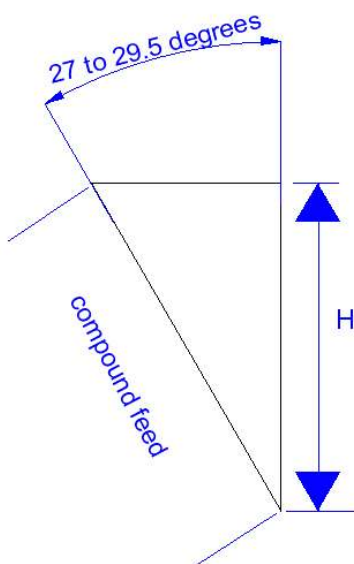
The Non-standard Case (just for doc)



If you can't be bothered turning the nominal diameter down to the major diameter nor blunting the cutter tip by $\frac{H}{4}$, then this equation is for you:

$$\text{Total compound infeed} = \text{pitch} = \frac{1}{TPI}$$

The compound should be set over at an angle between 27 and 29.5°.



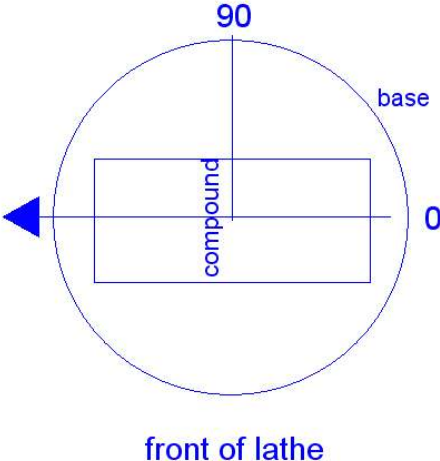
Don't forget to reduce the nominal diameter to the major diameter when done cutting the thread. That will be a reduction in **diameter** of $\frac{H}{4} = \frac{0.217}{TPI}$.

For example, if you are cutting 20 TPI, then the total compound infeed is $\frac{1}{20} = 0.050''$. Then remove $\frac{0.217}{TPI} = \frac{0.217}{20} = 0.011''$ from the diameter.

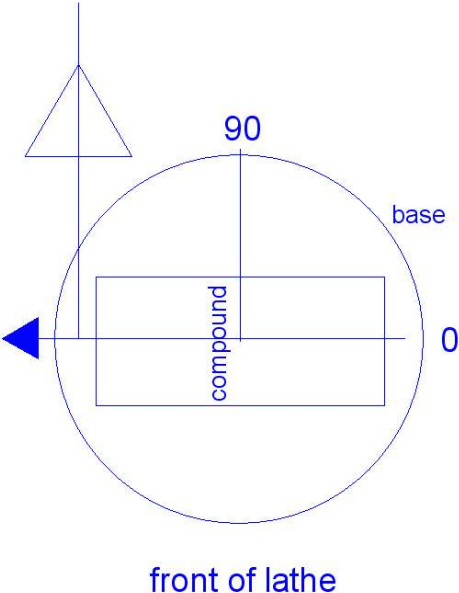
This non-standard root will be more likely to fail. But for many applications, this is not an issue because the part is not stressed that much.

The Malcolm Method

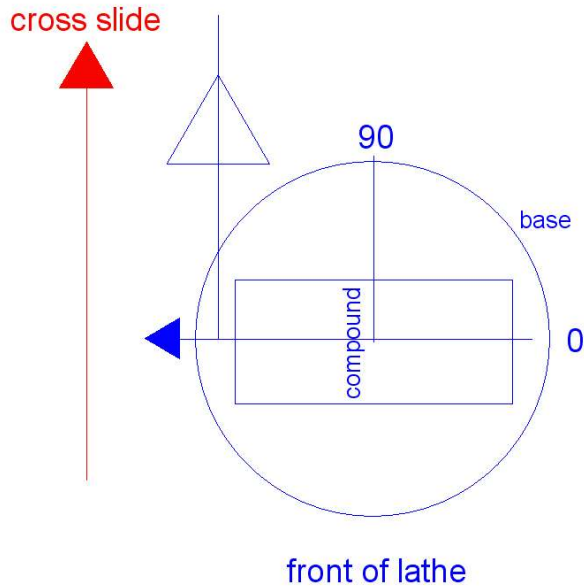
I do not know who invented this method but my friend Malcolm was the first to tell me about it. The advantage of this method is you can directly read thread depth on the cross feed dial as you are cutting.



The compound is set parallel to the longitudinal axis of the lathe. Movement forward is towards the headstock.

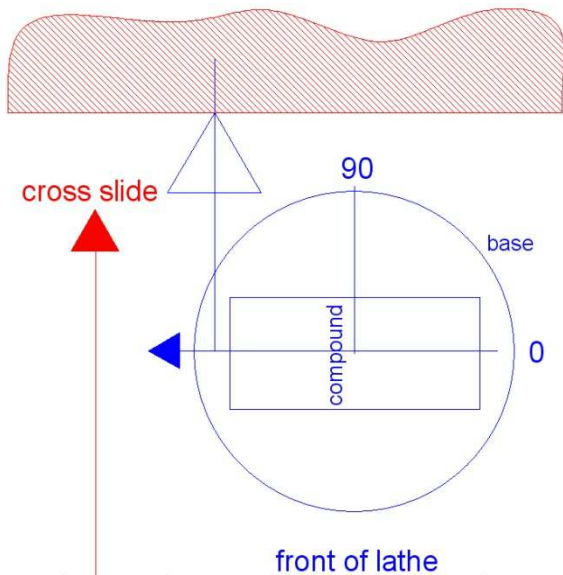


The cutter is attached to the compound. The centerline of the cutter is set perpendicular to the lathe's center of rotation.



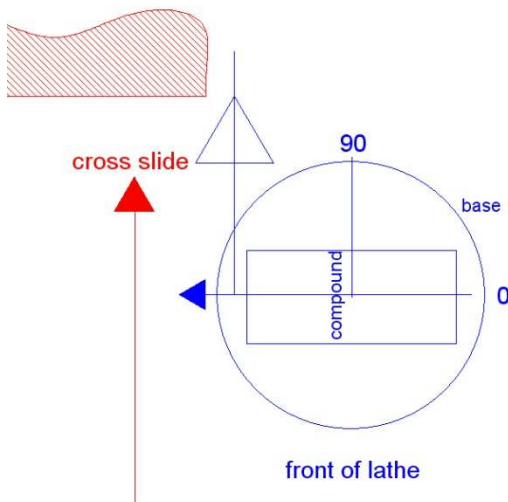
Turning the cross feed dial, I am able to feed the cutter into the part. Advancing the compound dial moves my cutter towards the headstock. These two motions are independent.

Contrast this action with the Traditional Method where the compound is set to around 30° . Feeding in on the compound moves the cutter *both* into the part and towards the headstock.



We will be feeding the compound towards the headstock so start by getting some room to move. Back the compound away from the headstock as far as it will go and then feed forward enough to take out the backlash. Zero the compound feed dial and then feed in $0.001''$. Starting at $0.001''$ makes the math come out nicely.

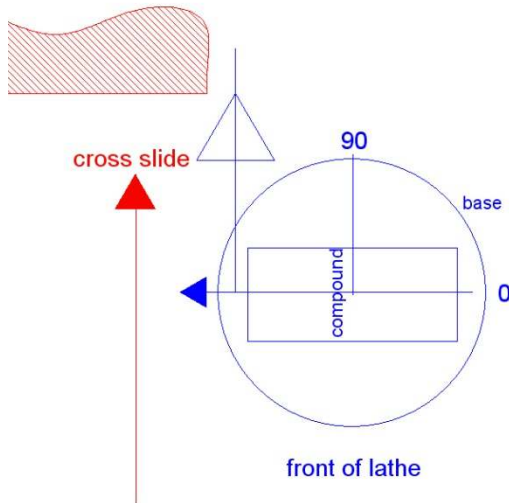
Next, feed in the cross feed until it touches the surface of the cylinder. The cylinder should have the nominal diameter defined on page 1. Zero the cross feed dial.



Back the cross feed out enough to clear the cylinder. Then move the carriage to the right so you are beyond the right end of the cylinder.

Turn the cross feed dial back to zero and then advance it by 0.002" in preparation for the first pass. We now have the cross feed dial at 0.002" and the compound dial at 0.001".

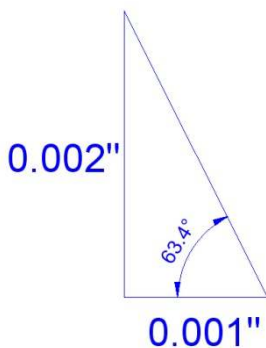
Start the lathe and make your first cut using the leadscrew feed. At the end of the pass, disengage the split nut, back the cross feed out enough to clear the cylinder, and return to the position to the right of the part.



We are now ready to take our second pass. Turn the cross feed dial back to zero and then advance it by the original 0.002" plus another 0.002" for a total of 0.004".

Advance the compound dial by 0.001" to 0.002".

We now have the cross feed dial at 0.004" and the compound dial at 0.002".



By accomplishing these movements, we end up moving the cutter at a 63.4° angle with respect to the longitudinal axis: 0.001" towards the headstock and 0.002" into the part. This is the same as setting the compound at $90^\circ - 63.4^\circ = 26.6^\circ$ and feeding in 0.00224".

Make your cut.

Again, back the cross feed away from the part and move all the way to the right.

We are now ready to take our third pass. Turn the cross feed dial back to zero and then advance it 0.002" beyond the last cut. The cross feed dial should now read 0.006".

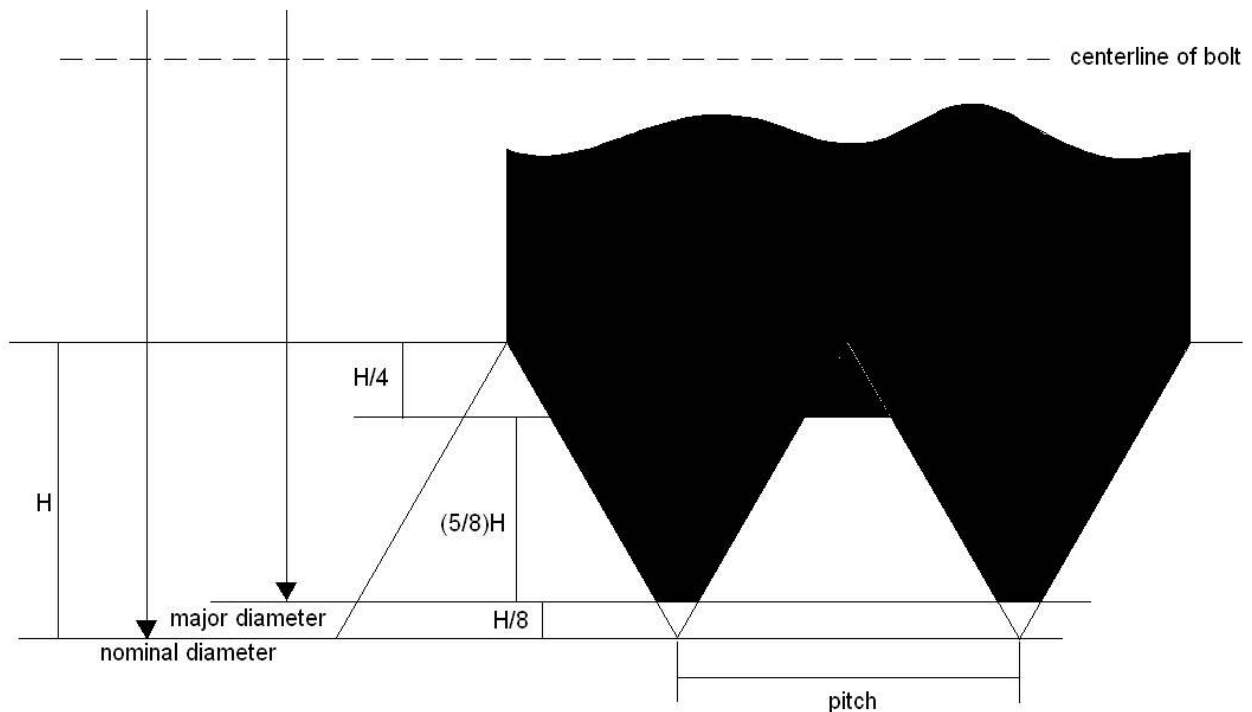
Advance the compound dial 0.001" beyond the last cut. The compound dial should now read 0.003". We have move 0.001" closer to the headstock.

Make your cut.

So the pattern is to feed in the cross feed 0.002” per pass and compound 0.001” per pass. Note that the cross feed dial reading always equals twice the compound dial reading.

As long as the cross feed change is twice the compound feed, you will maintain this in feed angle. So, for example, to take a cut equivalent to the compound feeding in 0.004”, you change the compound by 0.002” and the cross feed by 0.00447”. I show the extra digits just so you can see that doubling the compound and cross side advances does double the effective in feed.

When the cross feed dial reads $\frac{3}{4}$ of the thread depth, which is 0.65 times the pitch⁴, you are done cutting the thread. This does assume that the tip of your cutter has a radius that is back $\frac{1}{4}$ of the thread depth. That radius gives you the profile at the root of the thread.



The last step is to reduce the diameter of the thread down to the major diameter. This takes off that sharp edge. Assuming you started at the nominal diameter, you come down by $H/8$ in radius which is $H/4$ in diameter.

⁴ Recall that the thread depth, H approximately equals 0.866 times the pitch and the pitch is 1 divided by the threads per inch. So $H \approx 0.866 \times 0.75 \times P = 0.65P$.

That Professional Look and Feel



As a finishing touch, it is good to form a taper on the end of the threaded part. I've always done this on my belt sander but CT2 of gingery_machines pointed out that cutting the taper on the lathe with the threading tool gives better results.

Bench Work

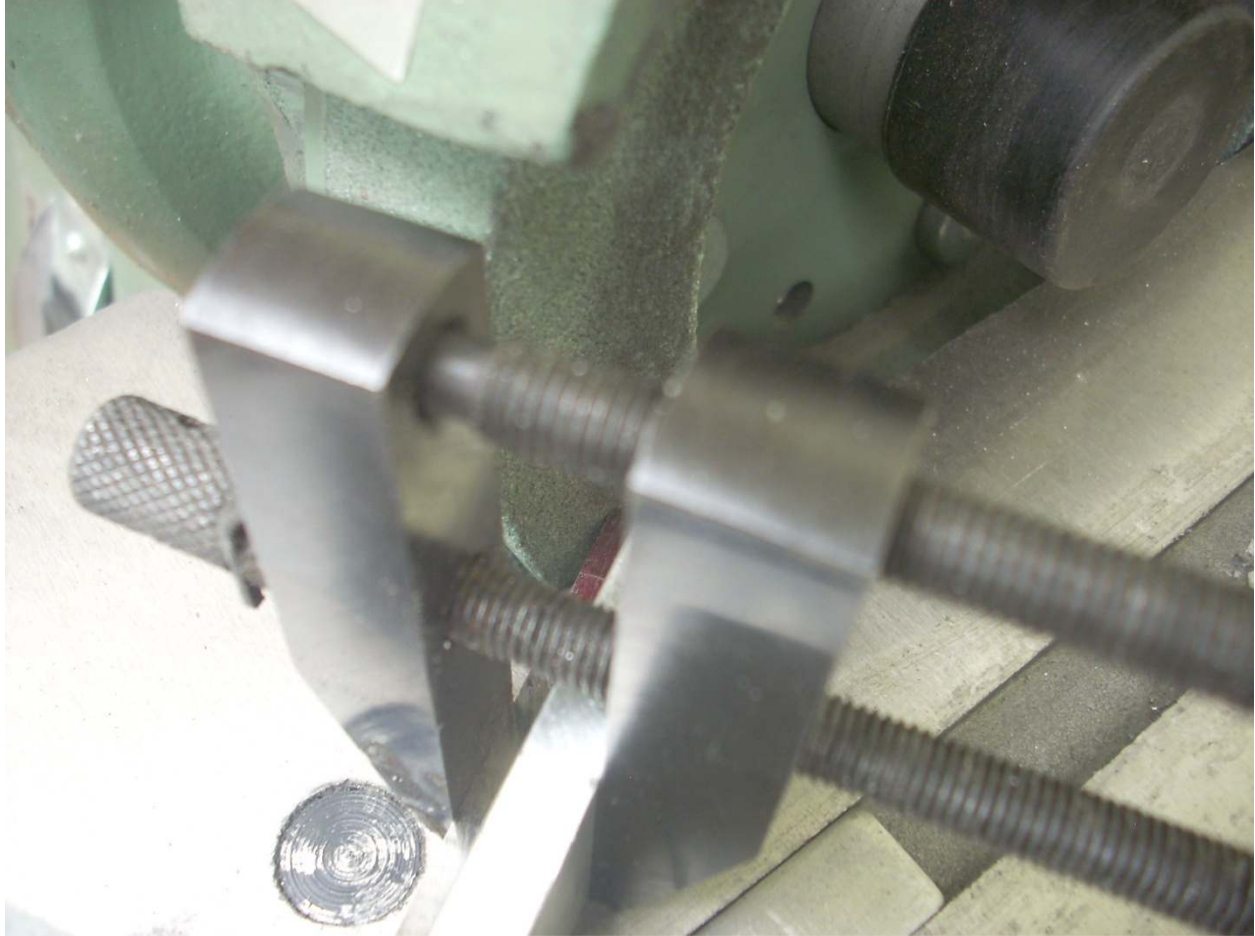
Here is where I put up or shut up.



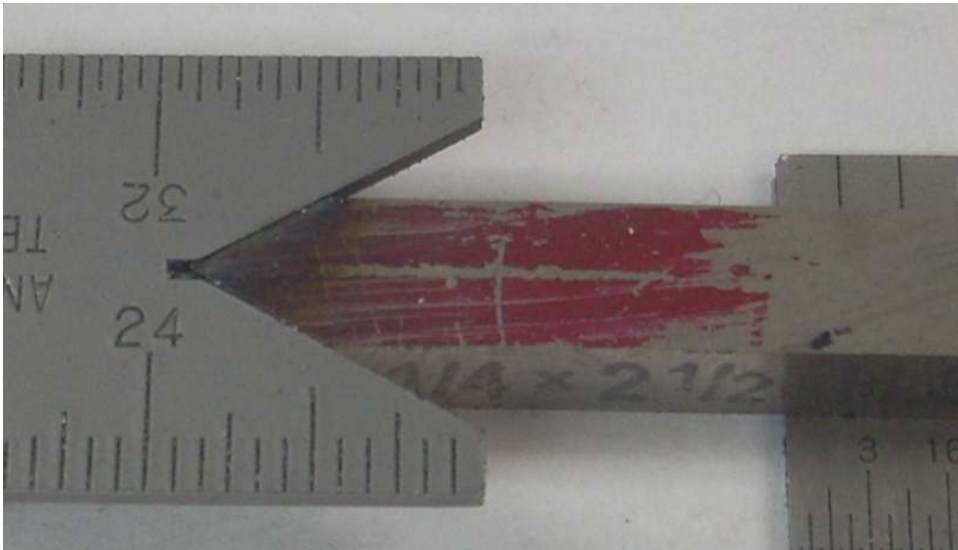
I first ground a cutter from $\frac{1}{4} \times \frac{1}{4}$ HSS. Rough layout lines were scribed through Dyken. Then I removed most of the metal on my bench grinder.



Next I set up my new tool grinder for exactly 30°. I'm using my "fish" to set the compound. The table has been tilted down 5° to give side relief.



The cutter is clamped to my aluminum angle which is pressed up against the compound. I'm taking a series of light passes. The set up is then flipped over and my fish again used to set 30° for the other flank.



This is a close up of how well the cutter fits the fish. No light shines through the interface.

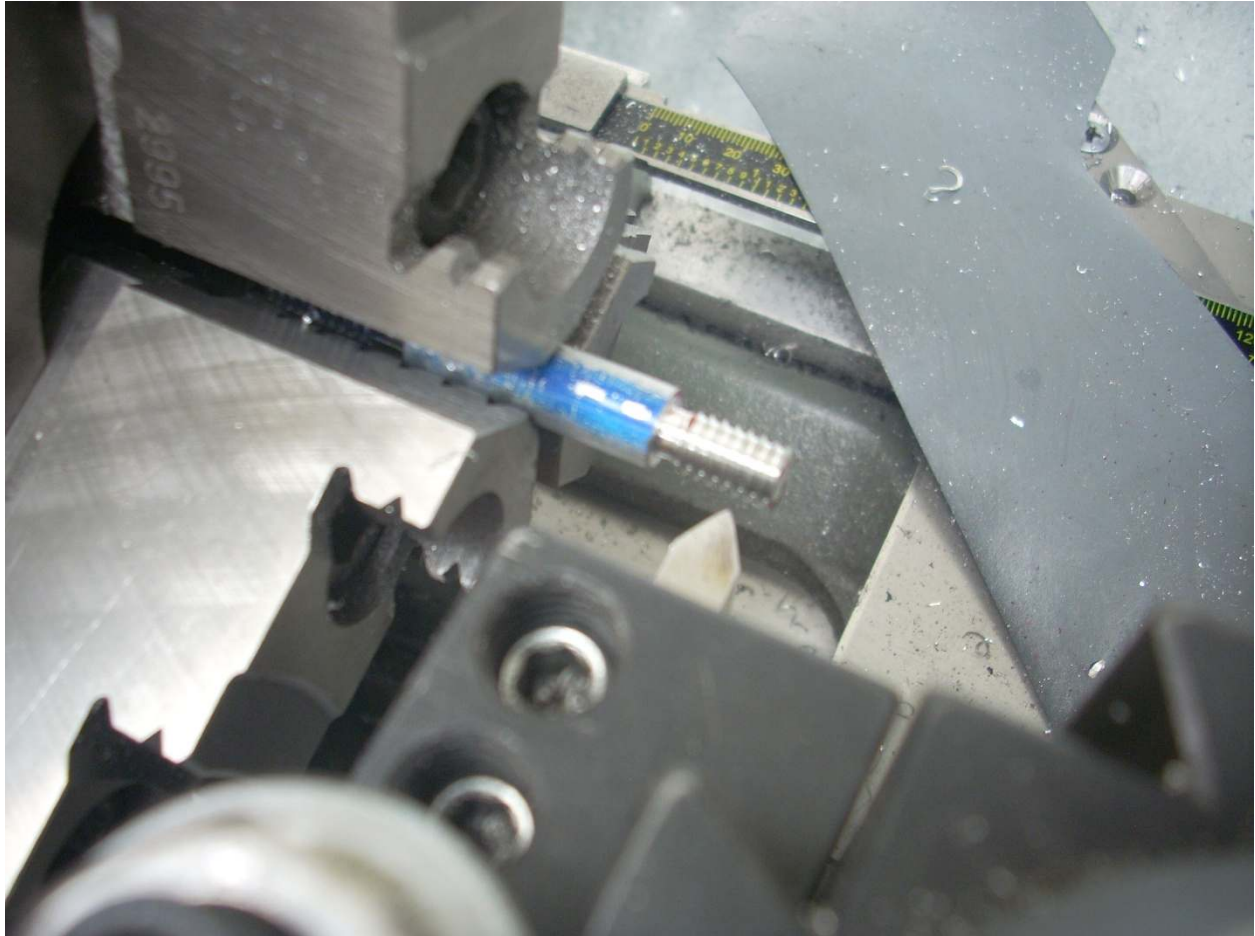


I'm using my calibrated loupe to put a 0.01" flat on the tip. It took about 20 passes with a stone. Ideally the flat should be $\frac{1}{4TPI} = 0.0125"$.

I was unable to use Malcolm's trick with the mic because this size HSS does not

reach across the spindle's diameter.

I first turned the outside diameter to the major diameter. Machinery's Handbook says for a 1/4-20 thread, 1A class, the major diameter should be between 0.2367" and 0.2489". Using my wiz bang digital mic, I got a diameter of 0.24850" so I am 0.0004" below the maximum and 0.0118" above the minimum.



The cutter was placed in a Quick Change Tool Holder and set to the height of the center of rotation. Then I used my fish to align the center line of the cutter to the perpendicular of the part. Here you see that the thread is almost done.

The equation says that I should feed in the compound $\frac{0.619}{TPI} = \frac{0.619}{20} = 0.0031''$. I made three passes at 0.005'', 3 passes at 0.004'', one pass at 0.003'', and one pass at 0.001''. There was a lot of spring in the aluminum part so I ended up making 7 more passes without advancing the cutter before the chips stopped coming up.

I then used the 3 wire method to measure the pitch diameter. Machinery's Handbook says that for a 1A class 1/4-20 thread, the pitch diameter should be between 0.2108'' and 0.2164''. I measured 0.2136'' while on the lathe and 0.2155'' with the mic in its fixture and easier access to the part. The average pitch diameter spec is 0.2136'' so I'm sure it was just dumb luck that I hit that number on the lathe.



Here is one close up of the thread before I cut the bevel on the end.



This picture was taken through my loupe. You can see some chattering on the root but the thread flanks aren't too bad.

You can also see the nice bevel made by using the left flank of the cutter. I just fed in with the compound.

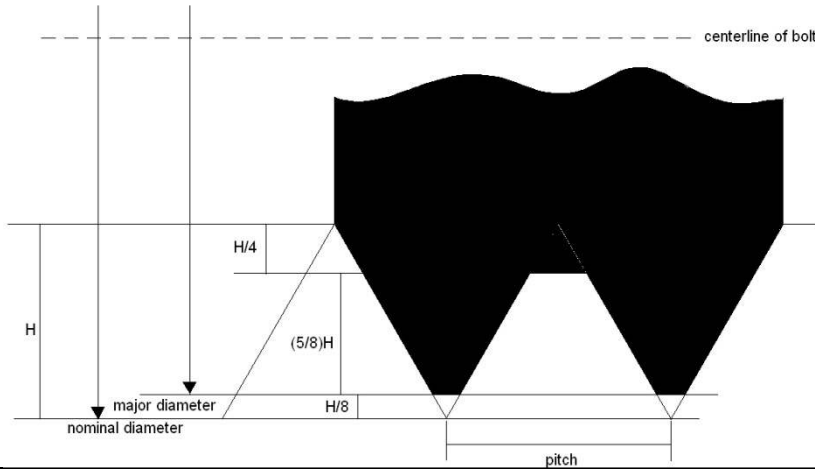
As a final test, I ran a nut on the thread. It went on easily without much play.

Reminder Card

The following page contains the essence of this article and can be printed out as a reference sheet.

Single Point Threading

All equations are in inches.



Parting Tool Method: Total cross feed			
Starting dia. ↓	Cutter tip →	Sharp	Blunted by
			(0.217)/TPI
Nominal diameter		(0.866)/TPI	(0.65)/TPI
Major diameter		(0.65)/TPI	(0.541)/TPI

Compound Feed Method: Total compound in feed			
Starting dia. ↓	Cutter tip →	Sharp	Blunted by
			(0.217)/TPI
Nominal diameter		1/TPI	(0.75)/TPI
Major diameter			(0.619)/TPI

Malcolm's Method: Total Compound; Total Cross Feed			
Starting dia. ↓	Cutter tip →		Blunted by
			(0.217)/TPI
Nominal diameter			(0.375)/TPI; (0.75)/TPI

Notes

The cutter must have an included angle of 60°.

A cutter blunted by $\frac{0.217}{TPI}$ has a flat on the end $\frac{1}{4TPI}$.

If you cut the thread starting at the nominal diameter, you must remove $\frac{0.108}{TPI}$ from the radius in order to get the major diameter.

Acknowledgements

Thanks to Donald of gingery_machines for pointing out the problem with the Parting Tool Method. Thanks to Walter of atlas_craftsman for helping me understand the effect of setting the compound at less than 30° . Thanks to Russ of atlas_craftsman for explaining how the Parting Tool Method can cut a wandering thread. Thanks to CT2 of gingery_machines for seeing that the compound's feed tells you thread depth when using the Traditional Method, mentioning an alternate way to cut the taper on the end, and how to avoid a burr as the nominal diameter is reduced to the major diameter. Thanks to doc of atlas_craftsman for pointing out why you don't get a wandering thread with the Parting Method, helping me solve the mystery of the compound set over value, and questioning the thread profile plus checking overall clarity of the article. Thanks to Bill of atlas_craftsman for explaining the effects of setting the compound over more than 30° . Thanks to Russ Kepler for testing the effect of compound angle and taking the picture. Thanks to Bruce of atlas_craftsman for suggesting that I include the tip width equation. Thanks to Chris for noticing that most of the figures were up-side-down. Last but certainly not least is Malcolm from gingery_machines for his patience as he carefully explained his method to me.

These generous people again demonstrate that "all of us are smarter than any one of us".

I welcome your comments and questions.

Rick Sparber
Rgsparber@aol.com