A Toolpost Attachment for a High Speed Rotary Tool

By R. G. Sparber

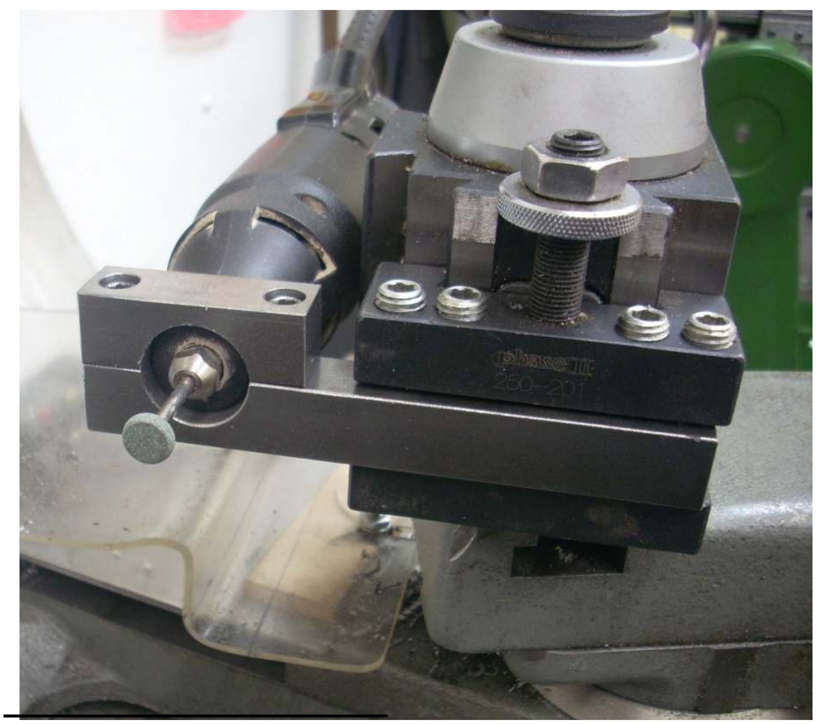
Copyleft protects this document[1].

Scope

This article is the second in a series aimed at helping someone new to metal working. I will show you how I go from rough idea to finished product including a few dumb mistakes along the way. As I have gained experience over the years, I still seem to make plenty of dumb mistakes.

If you are an experience metal worker, I welcome you to read this article too. Send me a list of what I did wrong and I will include your comments in subsequent versions of the article.

The Finished Product



A High Speed Rotary Tool is downright handy at times. And it can be made even more useful if solidly supported in a lathe tool post. Recently I heard this called a "live cutter".

I won't go into the dangers of having a high speed cutter or abrasive wheel exposed like this but do be careful. My focus is just on

[1] You may copy and distribute this article but not change it.

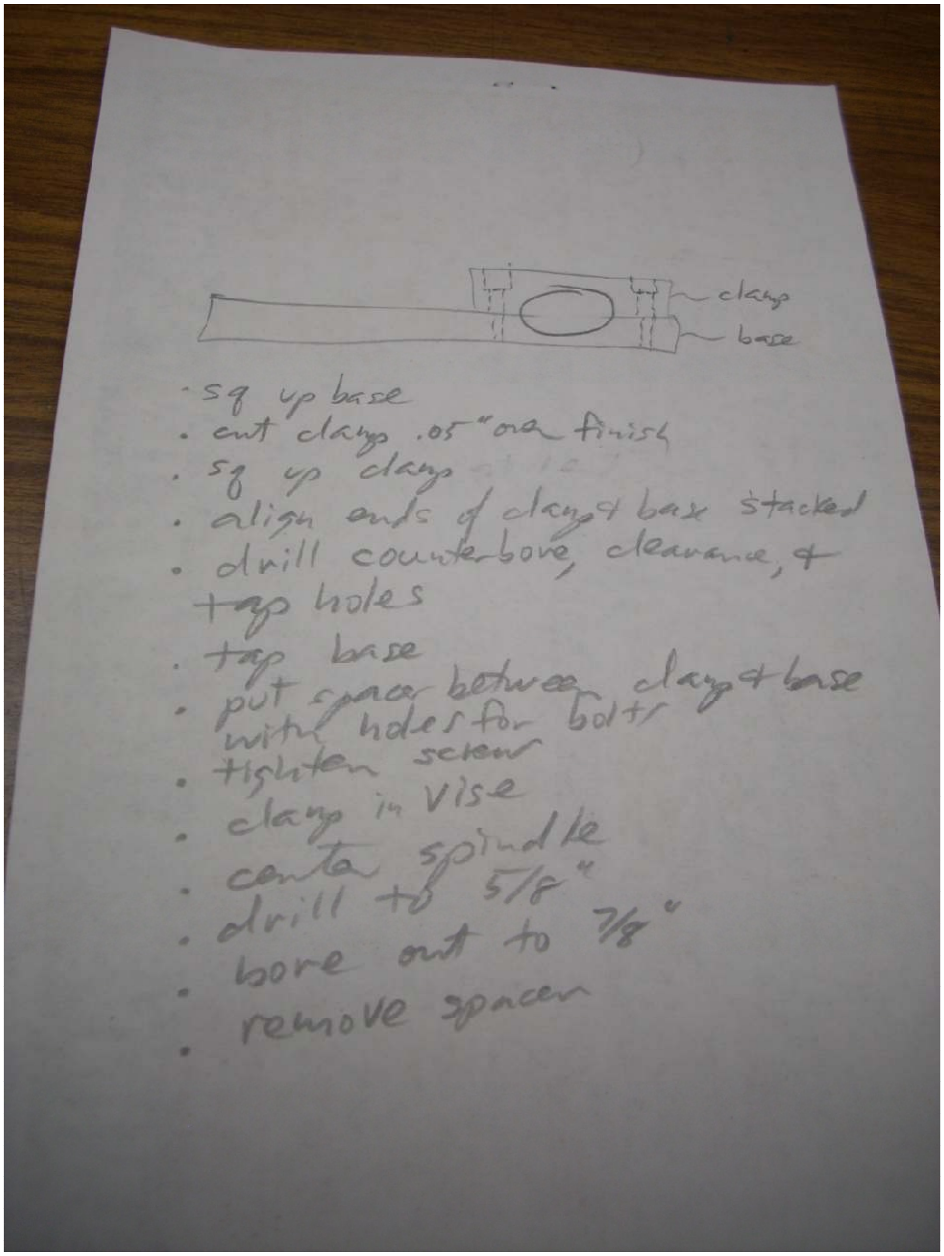
making this holder. The holder is made from two pieces of ½" x ½" cold rolled steel (CRS) attached together with a pair of socket head screws. I will call the long bar the support and short bar the clamp.

The Starting Point

I became interested in this project after reading about a similar tool for sale on eBay®. Beyond the fact that I'm not about to pay someone else to have fun making a tool, I also saw that it was designed for a Dremel® tool that had a threaded nose. As you can see here, mine just has an exposed metal ring. Truth be told, this is actually a Craftsman® version of a real Dremel.

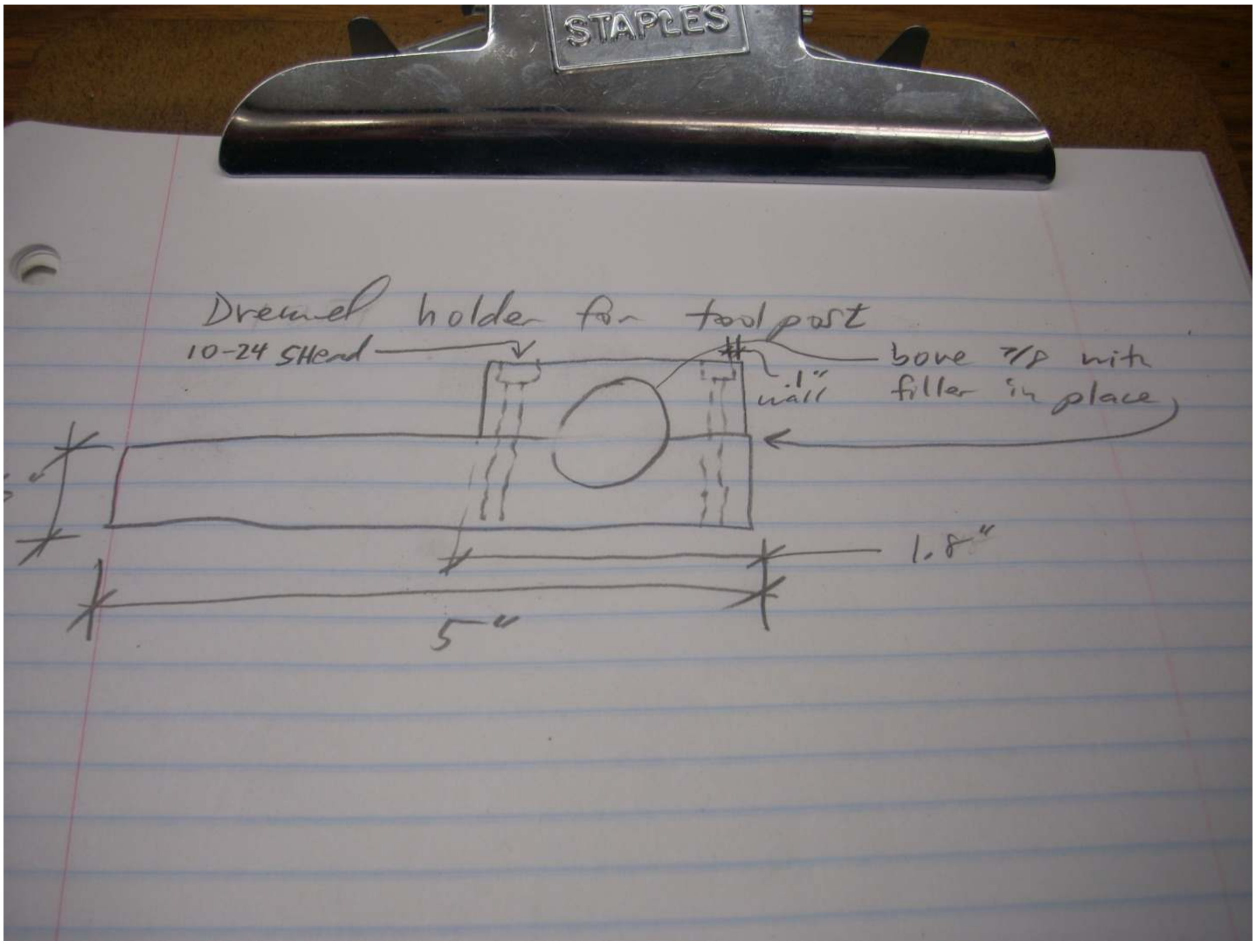
Having been properly inspired, I went off to my scrap bin to find suitable material for the project. Given the size of the hole I would have to make in the bars, I felt that aluminum would be too weak. Fortunately, I found two lengths of CRS that would do the trick. Both are ½" x ½" x 5". I also found some 1" long socket head screws in my selection of left over fasteners. The last necessary material is a strip of sheet metal about 0.03" thick. Its part in this project will become clear later.

Always Have a Plan



Once I had the materials in hand, it was time to capture the design in my head on paper. I can't tell you the number of times I have had a fantastic design in my head only to realize it could not work when expressed with paper and pencil. My plan is nothing fancy but attempts to hit the key points. I have a sketch of the design at the top. Below it are the steps needed to make it. Space is left between the steps because I often forget things that need to be done. Unless the project is extremely simple, I try to always have a plan. The plan will change as I proceed but still provides an essential framework.

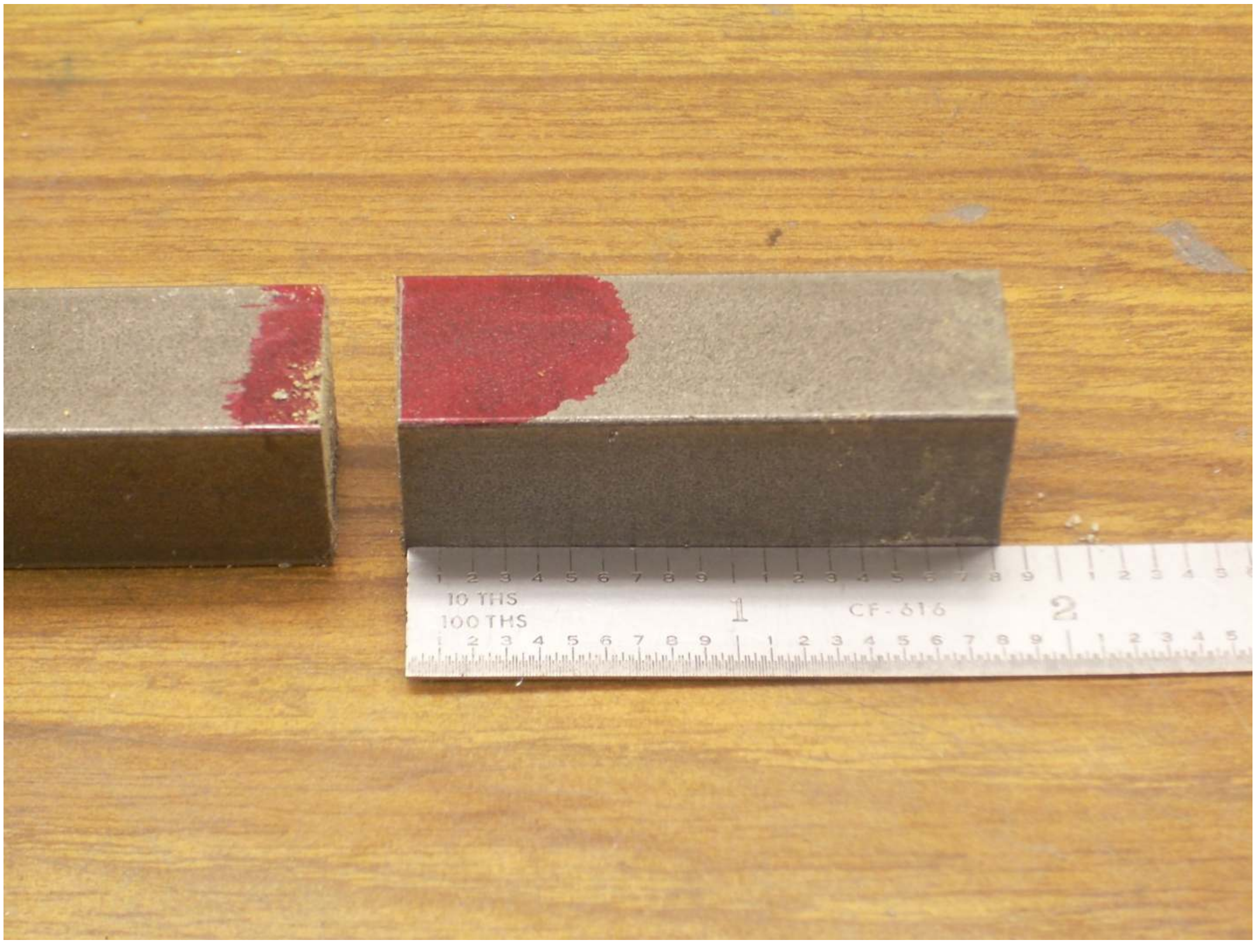
Shop Notes



Once in my shop, I add details to my original sketch. It is still nothing fancy but does the trick. This sketch is part of my running shop notes. As I proceed, missing dimensions are added to this and subsequent drawings. I show all of my calculations both to expose errors later on and to make it easy to see numbers necessary in subsequent steps.

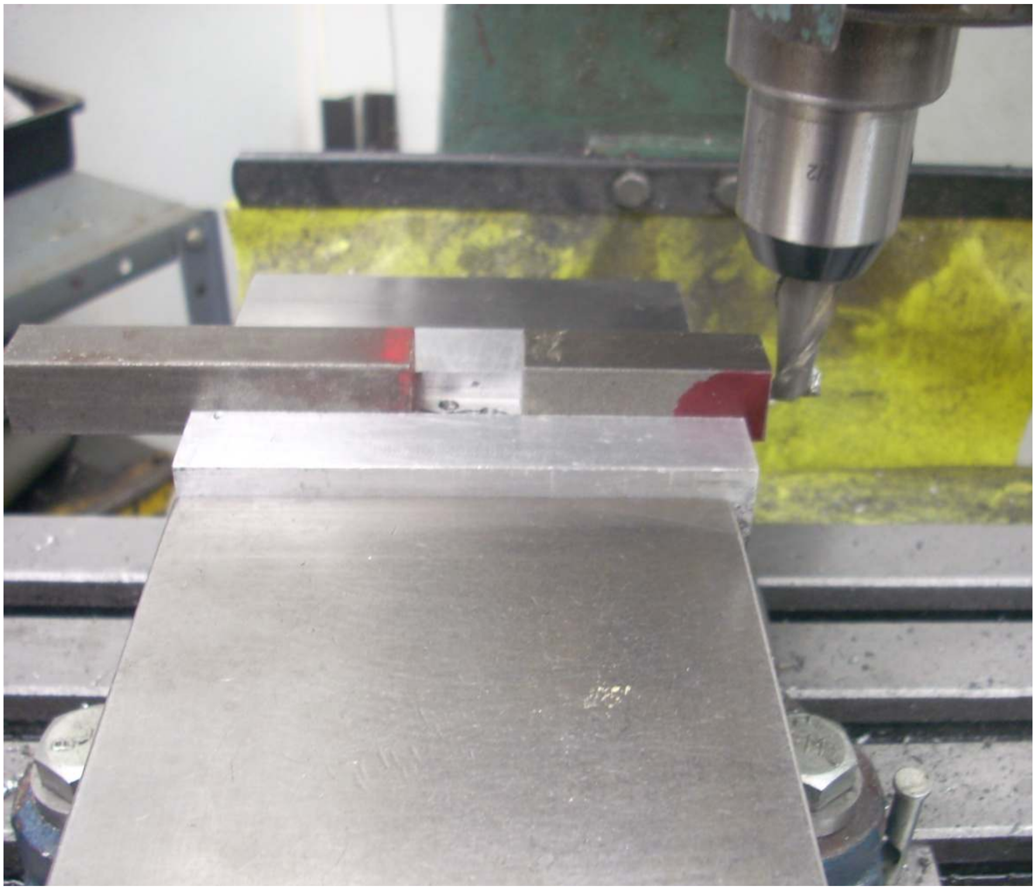
Some design work is going on here. I want to have 0.1" of metal between the end of the clamp bar and the start of the hole for the fastener. The fastener head is a little less than ¼" in diameter and ¼" tall. The High Speed Rotary Tool has a diameter of 7/8” and I do not want this hole to cut into the fasteners. A little back of the envelope figuring told me that my clamp bar should be 1.8” long. I added 0.05” to that for the rough cut so there would be room to true it up on my mill.

Rough Cutting of the Clamp Bar



I used my tenth inch ruler to get me close to the 1.85” line. I then used red Dykem® marking fluid to color the area. Then I used a small square and a sharp knife to draw a thin line through the red dye at the desired position. I could have done the measuring right on my bandsaw but wanted you to see the value of marking fluid. After the piece was sawed off, I used my belt sander to remove all burrs from both pieces.

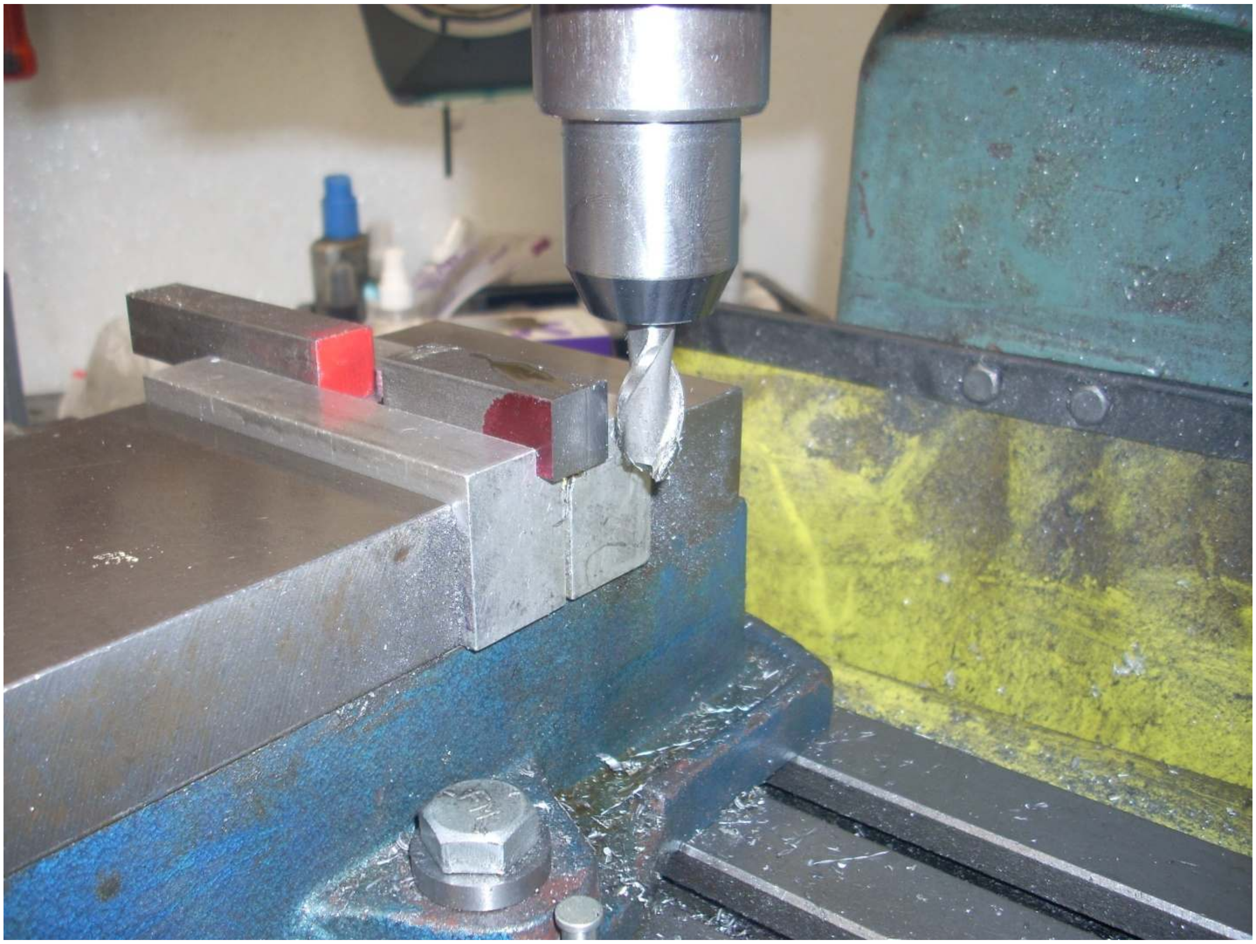
Truing Up the Clamp Bar



You are looking at my mill vise which has steps cut into the top edges to hold the part. On the right is our clamp bar with the sawn face next to my end mill. On the left is the rest of the bar. It is being used as a spacer so the movable jaw of my vise doesn't twist.

Note that my end mill is being held in an end mill holder. I prefer this arrangement over using a collet because it grips the end mill better without having to over tighten the draw bar. I have been told that a better quality collet would not have this problem.

I'm using cutting oil here. You may not be aware that there are two kinds of oil commonly found in a shop. One oil is for lubrication and permits metal to slide easily. The other oil is cutting oil. It prevents metal from sliding easily and therefore makes it easier for the cutter to grab the metal being cut and tear off tiny chunks of it.



The end of the clamp bar has now been trued up via side milling. I only took off around 0.01”. This is just enough to get the end square. I will not square up the other end now but wait until final assembly. In this way I will be able to cut both support and clamp bar at the same time to make them perfectly line up without having to precision measure them.

Truing Up the Support Bar

The support bar is long enough that I don't need to add in a spacer block.

You can better see the steps in my vise jaws which are made of 6061 aluminum. If this was to be precision machining, I would cut a few thousandths of material off of the horizontal and vertical surfaces with the end mill to true them up.

After milling the end, I deburred the edges on my belt sander. It is amazing how well these machines remove metal.

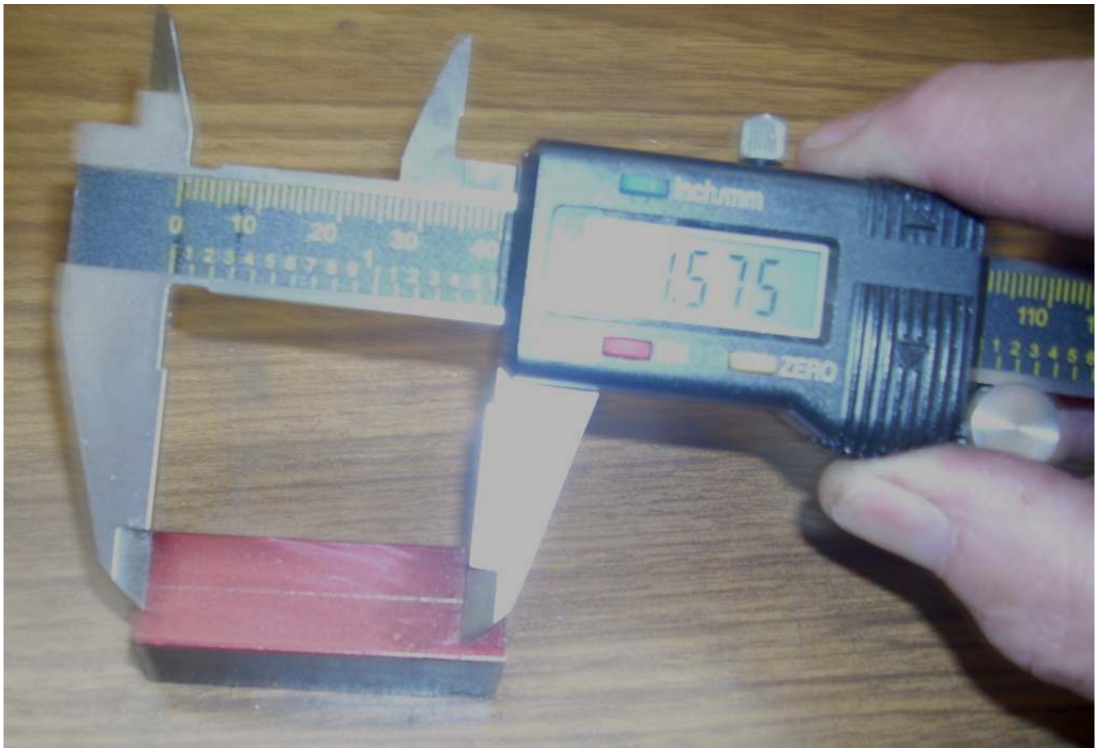
In the background you can see a wall in my shop. I use large heavy plastic sheets on the walls to prevent splattered oil from reaching the drywall. This is the same plastic used in some commercial bathrooms. The oil cleans up with a rag. The bright white finish helps to reflect light around the shop.

Laying out the Clamp Block

With the clamp block deburred, I first use my red dye on the surface. I then lay out my lines that will locate my two screw holes. To start, I measure the actual width of the block. Minor surprise here. I thought I was using ½" bar but it is actually closer to 5/8". It doesn't really change anything.

My Harbor Freight® digital caliper reads 0.622”.

In order to scribe my center line, I set it to around 0.311”. No need to be exact here because I will run it down both sides to define my center line.

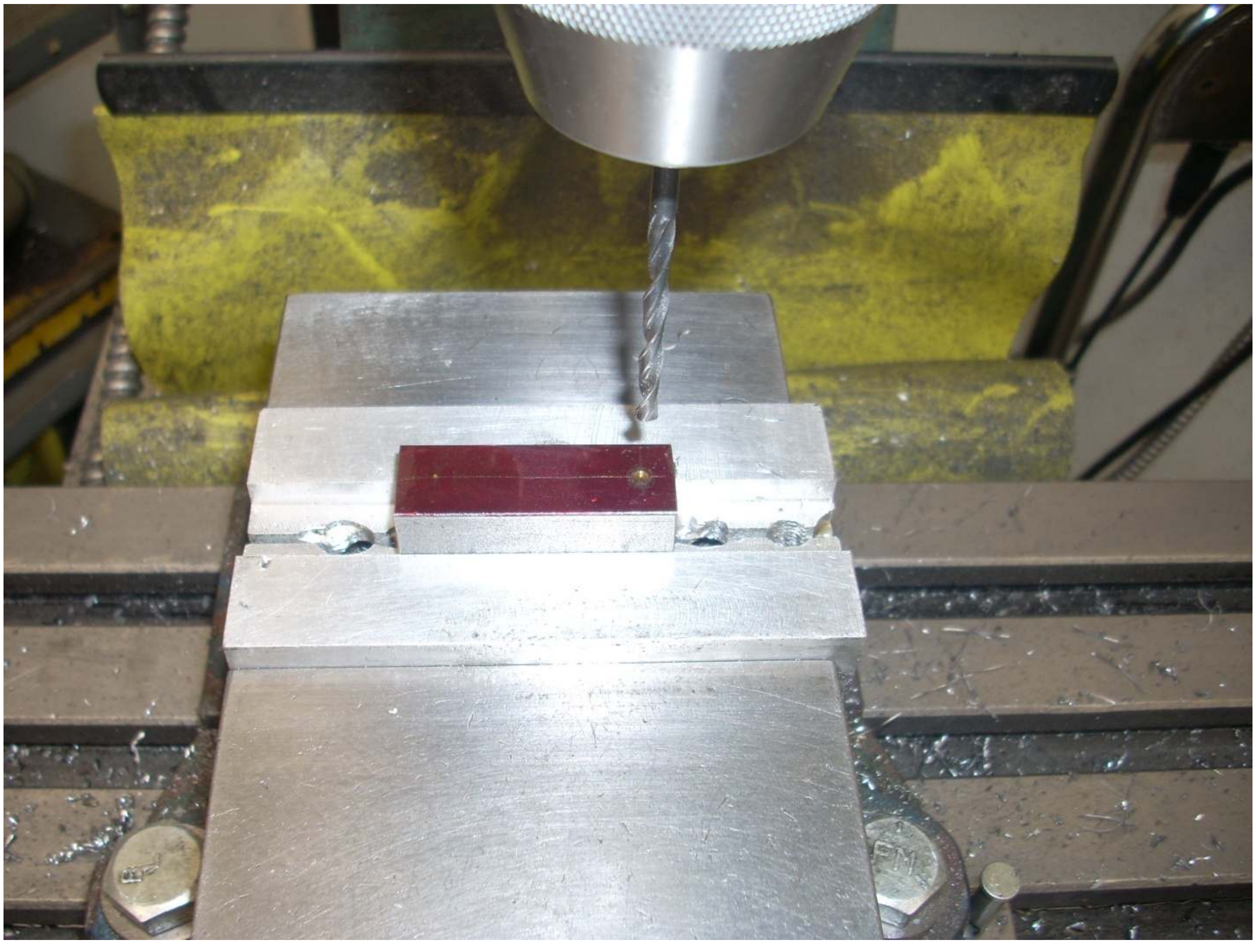


Measuring from the trued up end, I scribed a line at 0.225” which is my 0.1” plus the radius of the hole for the head of my screw (0.1” + 0.125”). The second hole is 1.575” from that same edge. Here you see my caliper set to this value. I will then put one blade against the trued up end and mark my line across the center line.

Note that I trued up one end and then use it for all measurements. I would only measure from each end if the overall length of the bar had been milled perfectly to its final value.

These scribe lines are a little hard to see so I am using a mark out spring loaded punch at each intersection.

Machining the Clamp Block



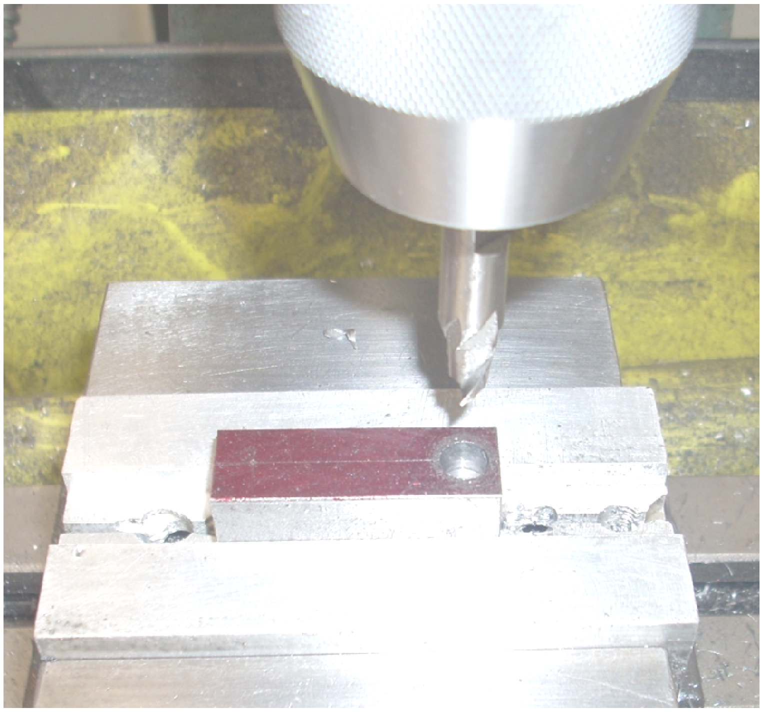
The clamp block is placed back in my vise. The fasteners have a 10-24 thread which means I need a #25 tap hole and a #9 clearance hole. Now, I really don't care if these holes are precisely located but I do want the holes in the clamp block to match up with the holes in the support bar. I have chosen a procedure to insure that this happens.

I start with a centering drill that makes a cone shaped hole. The point of this cutter can be easily lined up with the tiny dent made by my layout punch. The X and Y axes are locked. I then drill with the centering drill and follow with my #25 tap drill. Eventually I will use the larger diameter #9 drill but not yet.

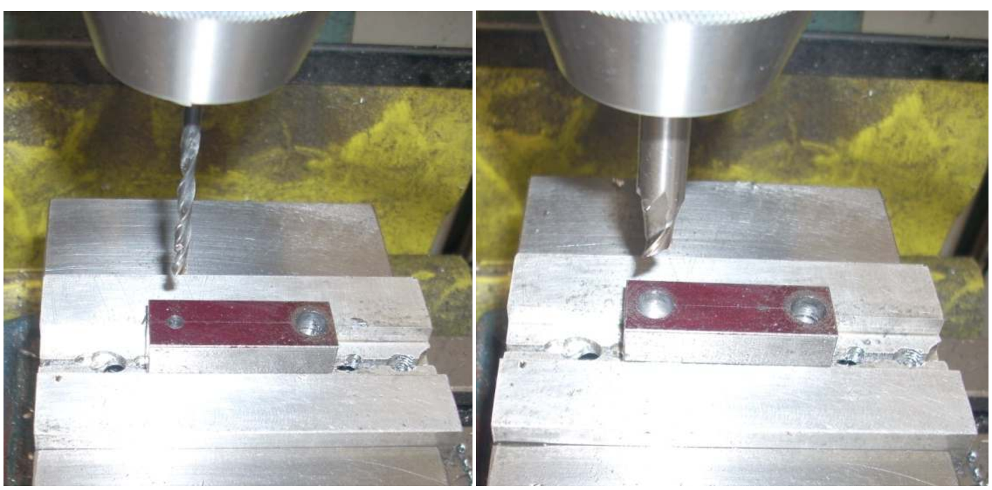
The next step is to drill a ¼" hole ¼" deep to accept the head of my screw. I want the bottom of the hole to be flat so am using a ¼" end mill in my drill chuck.

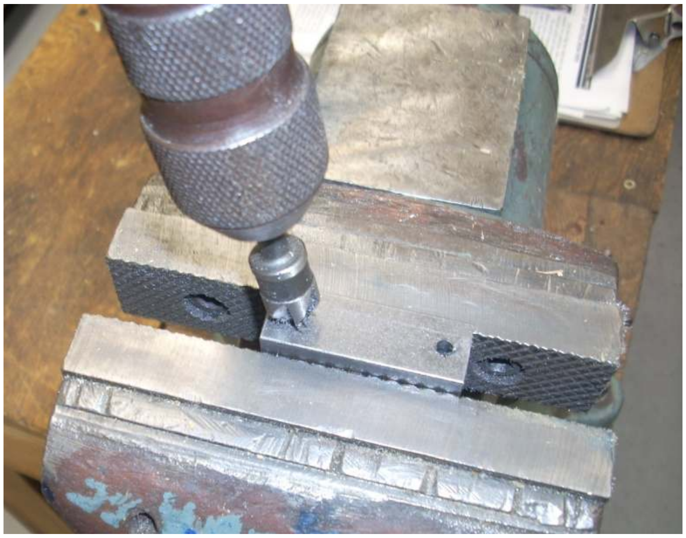
I am using a 2 flute end mill here which can be used to drill holes because it is able to "center cut". A 4 flute end mill cannot typically center cut so would not be a good choice for drilling holes. But since I already have the hole made with the #25 drill, I could use a 4 flute end mill here if I wanted.

I touch down the cutter without the mill running, set my zero, then raise it up, turn on the mill, and lower the cutter ¼" below the surface.



The process repeats for the second hole.

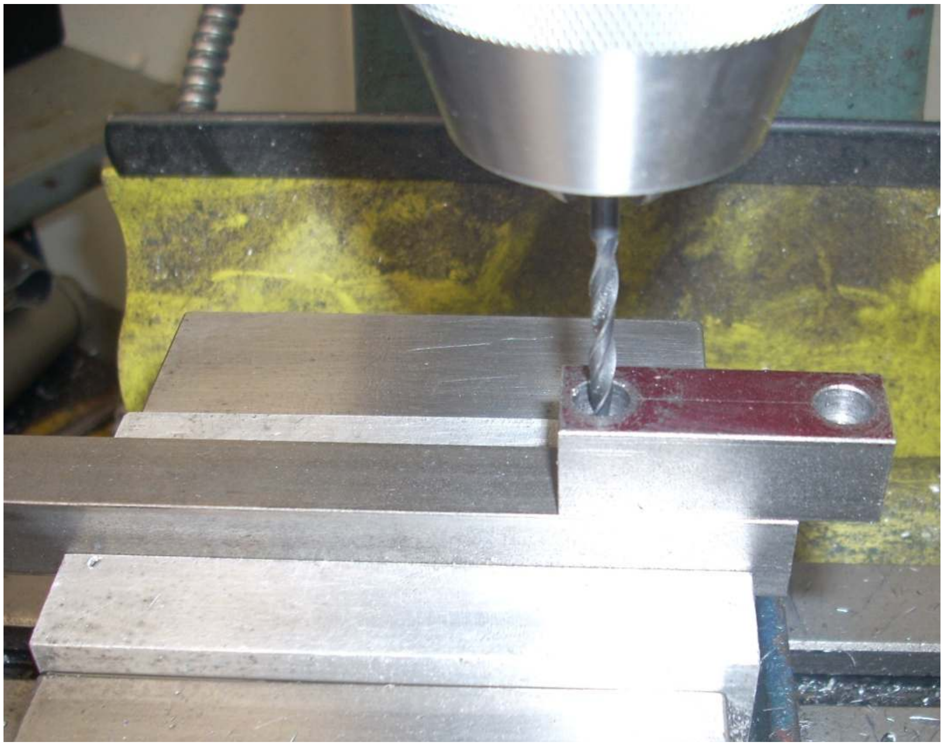




I then use a countersink mounted in a brace and bit to deburr both ends of each hole.

Here is the clamp bar so far.

Machining the Support Bar



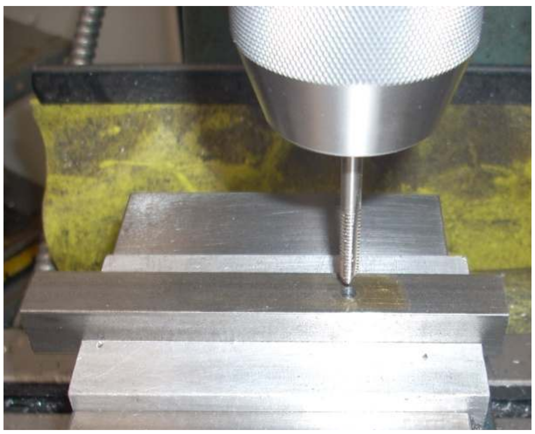
The support bar is put back in my vise. With my #25 drill in the chuck, I lower it part way down through the clamp bar. The mill table's Y axis is moved until the two bars are aligned front to back. I then move the X axis until the ends line up.

Here you see the two bars lined up. I then raise the drill, remove the clamp bar, and drill through the support bar.

All went well until, snap, there went my #25 drill! What happened? Well, it might have been a hard spot in the metal, a dull drill, feeding down to quickly, or breaking through the bottom of the hole. It might also be a combination of these problems.

After buying a new drill, I determined that it did break at the bottom of the hole so most likely was this and a dull drill and excessive feed. Happens to all of us…

The next step is to use a spiral point 10-24 tap. I chuck the tap into my drill chuck,



put plenty of cutting oil in the hole, and bring the mill up to speed. Then, with the tap just above the hole, I cut power and feed down. These spiral point taps are designed to just feed in and not be periodically turned backwards to break the chip as is necessary with a hand tap.

The tap goes in far enough to be secured in the hole. A tap handle can then be attached to finish the job.

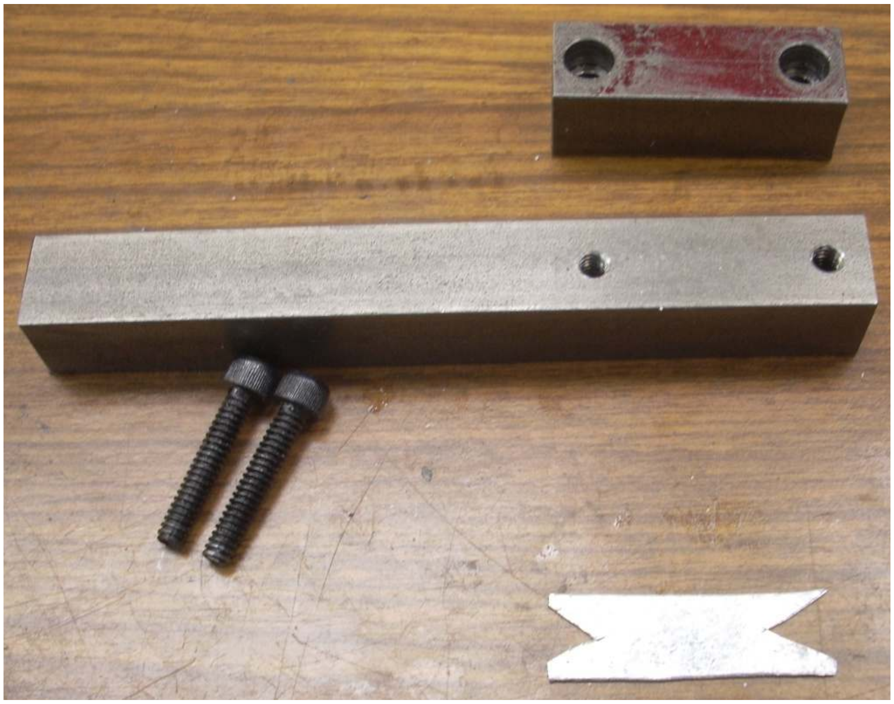
I can now open out the first hole in the clamp block to my clearance diameter. I don't want to disturb my mill so will use my Gingery drill press and homemade vise for the task.

Both ends of the hole are deburred with my brace and bit plus countersink.

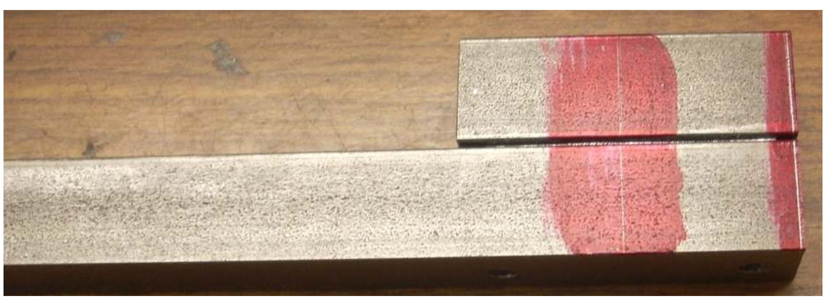
I can now use one of the screws to bolt the clamp block to the support bar. I then put in my #25 drill and move the X axis until it freely goes into the second hole. The clamp block is then swung out of the way and the second hole is made in the support block. I should have started the hole with a centering drill but cheated and just used the #25 drill. It worked OK but is not the best practice. The drill did wander a little before starting to cut.

Note that the first hole was completely finished including the fastener before starting on the second hole in the support block. The two holes must perfect align yet no measuring was done.

Boring the 7/8" Hole



In order to get a good clamping action on the nose of the High Speed Rotary Tool, I will put a strip of that sheet metal between the clamp block and support bar. I have notched it out so it will be captured by the screws. The bored hole will then have a gap the thickness of this sheet metal.



With it all assembled, I then used my Dykem again. I have marked the centerline of the hole to be bored plus how much to clean up at the right end. I originally planned to mill this end true but later decided to just use my belt sander.

Boring the Hole

It is time to move the head on my RF-30 mill/drill. Its height must be such that I can fit in my boring head as well as my 5/8” drill. Both are trial fit before the head is locked to the column.

Although not idea from a support standpoint, my boring head will be cutting with the quill all the way down.

In order to use my 5/8” drill, the quill must be raised up all the way. I have my ½” drill in the chuck here.

That red dot on the spindle marks the radial location of my internal alignment pin.

I’m using a pointed length of ¼” rod, called a spud, in my drill chuck. It makes it easy to align the spindle with the center of the hole to be bored out. The X and Y axes are then locked.

I’m now using my ¼” centering drill to provide a cone shaped hole for my first drill.

My first hole is ¼” in diameter. I then open it out to 3/8” and then ½”. This is called step drilling and is both easier on the drills and faster than just starting with a ½” hole. My final drill is 5/8”. I then switch to my boring head.

The boring bar is set to just touch the inside of the drilled hole. I want to bore a hole 7/8” in diameter which is 0.8750”. This hole does not have to be very precise but I need the practice so will do the best I can. This means being able to know when I’m at the correct diameter.

The first step is to have an accurate distance of 0.8750”. I own a set of spacer blocks that are good to within +/- 0.0001”.

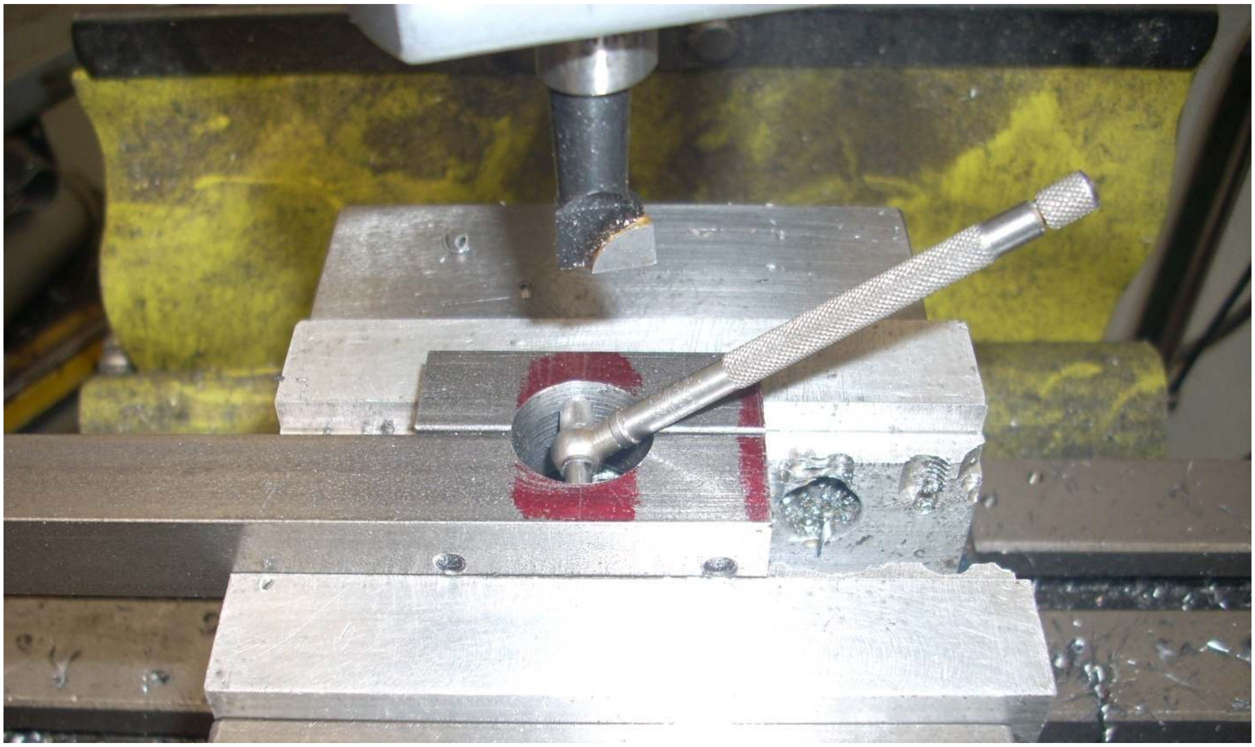
In order to get to 0.8750”, I first select a thickness that ends in 5. My 0.105” spacer does the trick. That leaves me 0.875” – 0.105” = 0.770”.

I then select a spacer that ends in 70. I have a 0.170” which leaves me 0.770” – 0.170” = 0.600”. I have a spacer 0.600” so am done.

Stacking my 0.600”, 0.170”, and 0.105” spacers up I get 0.875”. This is actually 0.8750” +/- 0.0003” since there are 3 blocks with each one accurate to +/- 0.0001”. I will refer back to this stack as I measure the bore. It is essential that all surfaces be spotless in order to achieve this accuracy.

The hole was drilled to around 5/8” and my finished diameter is 0.8750” which is a radius of 0.4375”. The boring head is advanced by a screw that reads out changes in radius so it makes more sense to talk about radius than diameter. Yet all readings are in diameter. In order to avoid confusion, I always place the symbol Φ next to a diameter reading and an “r” next to a radius in my shop notes.

While we are more than about 0.01” r away from our finished diameter, we don’t have to be too careful with our measurements. But just to illustrate how to use our spacer stack, I will use it to calibrate my caliper. I simply used the caliper to measure my stack and then zeroed it. In this way I am relying on the repeatability of the caliper and the resolution of the display to tell me when I am at my finished diameter. I will use the digital caliper until I’m within 0.01”r of finished so this is more than adequate for accuracy. I don’t normally use spacer blocks with a digital caliper.



The T shaped tool is a telescoping inside diameter measuring tool. The ends of the cross piece contact the bore and then can be locked in place by twisting the end of the handle. It takes a lot of practice and a light touch to get an accurate measurement. The telescoping cross piece is slid back and forth slightly to insure that I am really at a maximum reading. It is then locked and the test repeated. There should be no play between tool and bore. Then the tool is gently lifted from the bore and measured.

Once I am within 0.01”r of the finished bore, I switch to my micrometer.

The first step is to mic my spacer stack and record the reading. In this case I found that my mic read 0.0015” over 0.8750”. This means that if my mic reads 0.8765”Φ, my bore is actually 0.8765” - 0.0015” = 0.8750” within the accuracy of the spacer blocks and, more importantly, my skill to measure the bore.

Maybe it is just me, but I often get turned around when enlarging a hole. My solution is to write down every number and think about what it means. Otherwise I will approach the final radius with each pass of the boring bar and then just as carefully blow right past my target diameter making careful adjustments as I go.

My starting hole is around 0.312”r and my target is 0.4375”r. I advanced my boring head by 0.025” so would expect the new radius to increase by this amount and be 0.312”r + 0.025”r = 0.337”r. After the cut, I measured 0.340”r with my caliper. So my actual cut is 0.340”r – 0.337r = 0.003”r off from predicted. The first cut is typically off from the predicted value because there is uncertainty in the placement of the cutter plus the diameter of the hole cut by the drill. The hole might not even be perfectly round.

I am now 0.4375” – 0.340”r = 0.098”r from the target.

Advance another 0.025”, make the cut, and measure. This time I got 0.364”r which compares better with the predicted 0.340”r + 0.025”r = 0.365”r. The difference is most likely my measurement technique. Past experience has taught me that the advancement screw on my boring head is accurate to better than 0.001”. The hard part is seeing the advancement dial and moving it precisely.

With a radius of 0.364”r, I am now 0.4375”r – 0.364”r = 0.073”r from my target. I’ll make two 0.025”r passes before I measure again.

Now I measure 0.4145”r so have 0.4375”r - 0.4145”r = 0.0230”r to go. Advance the boring head by 0.013” and take a pass. The result should be a radius of 0.4145r + 0.013”r = 0.4275”r which is what I measured with my caliper. Beyond the fact that my caliper is only repeatable to 0.0005”, I am also very far from my target value. Yet Murphy’s Law is always with me. I get maximum accuracy when it doesn’t matter. I never hit the desired number on my finish cut.

I am now 0.4375”r - 0.4275”r = 0.010”r from my target. It is time to switch from caliper to mic.

Recall that when I mic’d my spacer stack, the mic read 0.0015” high. I can trust the mic to tell me how much to turn the boring head dial but my target value will be 0.8750”Φ + 0.0015” = 0.8765”Φ.

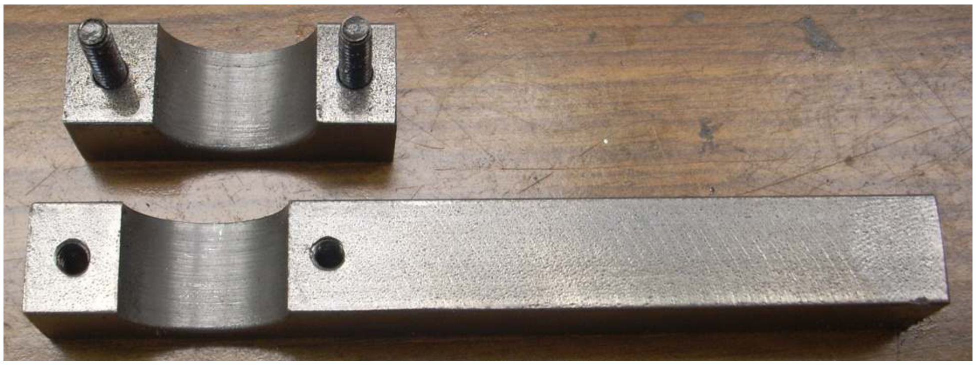
A light touch on the mic is essential. It should go without saying that all surfaces must be free of swarf or grit. This includes the bore.

I feed in 0.005” on my boring head and make my cut. Using the mic, I read 0.8684”Φ. This means that my diameter is really 0.8684”Φ – 0.0015” = 0.8669”Φ which is 0.4335”r. That is 0.4375”r - 0.4335”r = 0.0040”r from the target.

Using my magnifier, I do my best to turn the boring head dial in by 4 tick marks. The final pass is made and I measure 0.8769”Φ. This is really 0.8769”Φ – 0.0015” = 0.8754” Φ. So I ended up 0.8750”Φ - 0.8754” Φ = 0.0004” oversized in diameter. This accuracy is far better than what is needed for our project but there are plenty of situations where this would not be good enough.

Now, the above is all logical and correct. It is just not exactly what happened. Instead I took one more roughing cut than I should have and ended up at 0.8815” on my mike. This means a diameter of 0.8815”Φ – 0.0015” = 0.880”Φ which is 0.005”Φ ***over***. But since I was not writing down my proximity to my target, I got turned around and thought I had 0.0025”r to go. So I fed in another 0.0025” on my boring head and discovered I now had about 0.008”r to go. By this time it dawned on me that I had again been bitten by not following my own procedure. As I tell my students, “the lesson will be repeated until it is learned”. Crap.

Fortunately, being off 0.016" in diameter isn't fatal for this project. Recall that spacer I put between the base bar and clamp bar? It is about 0.03" thick. So I still have about 0.014" of gap.



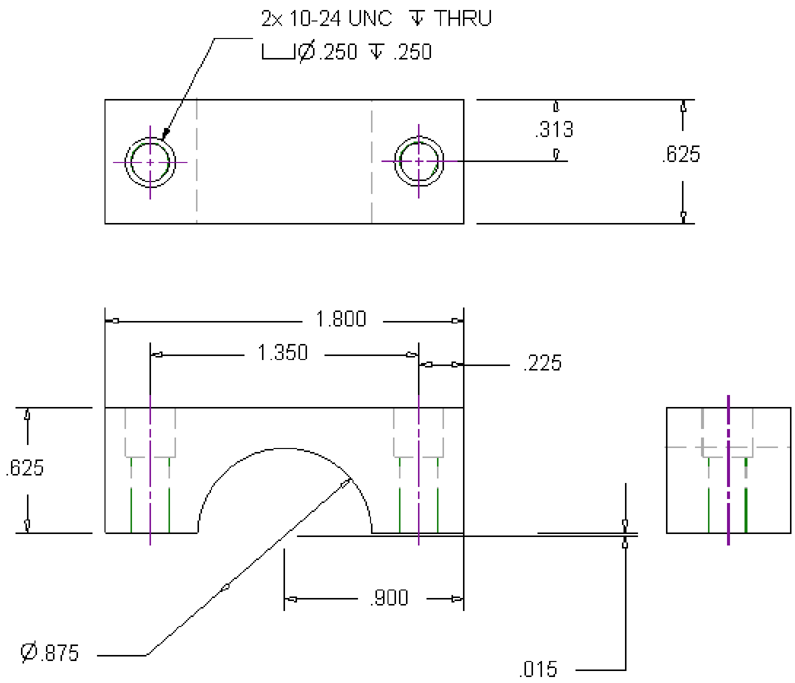
Here you can see the finished bore.

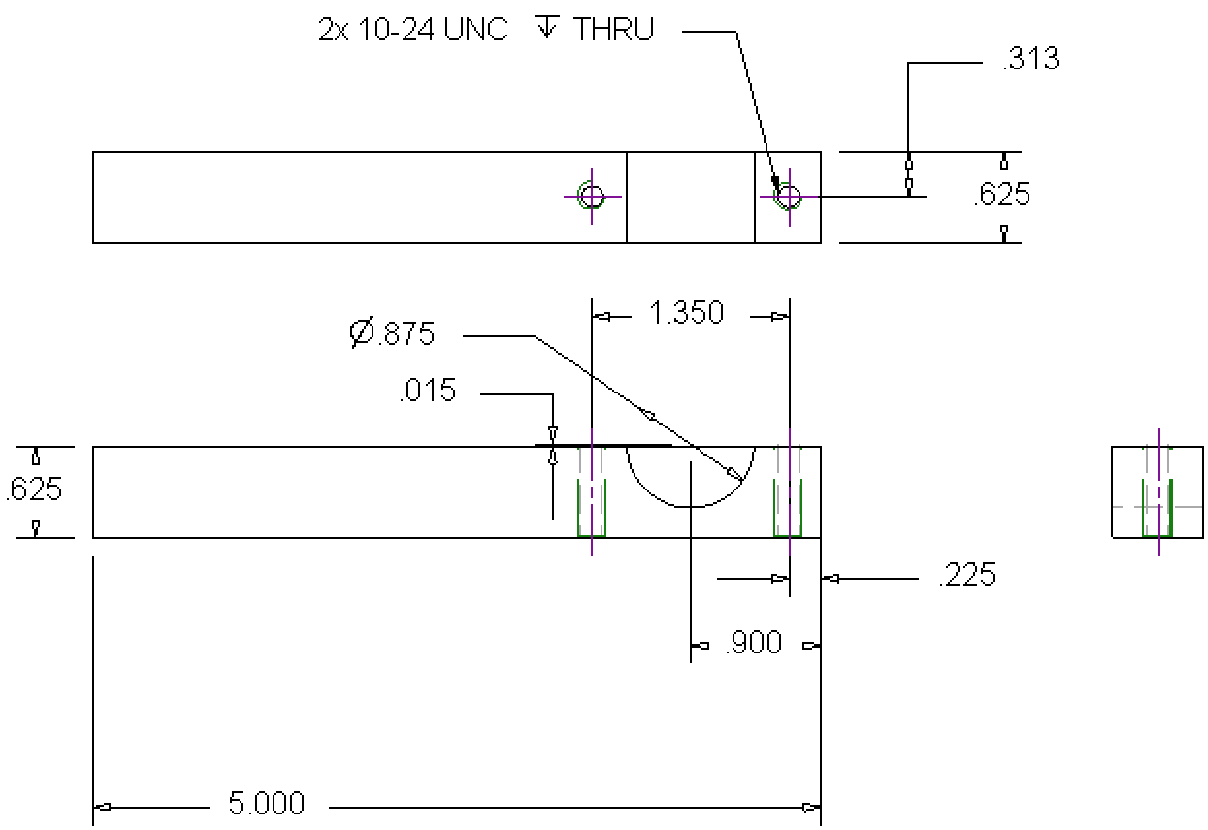
With the two pieces held together with the screws, I finished the end on my belt sander.

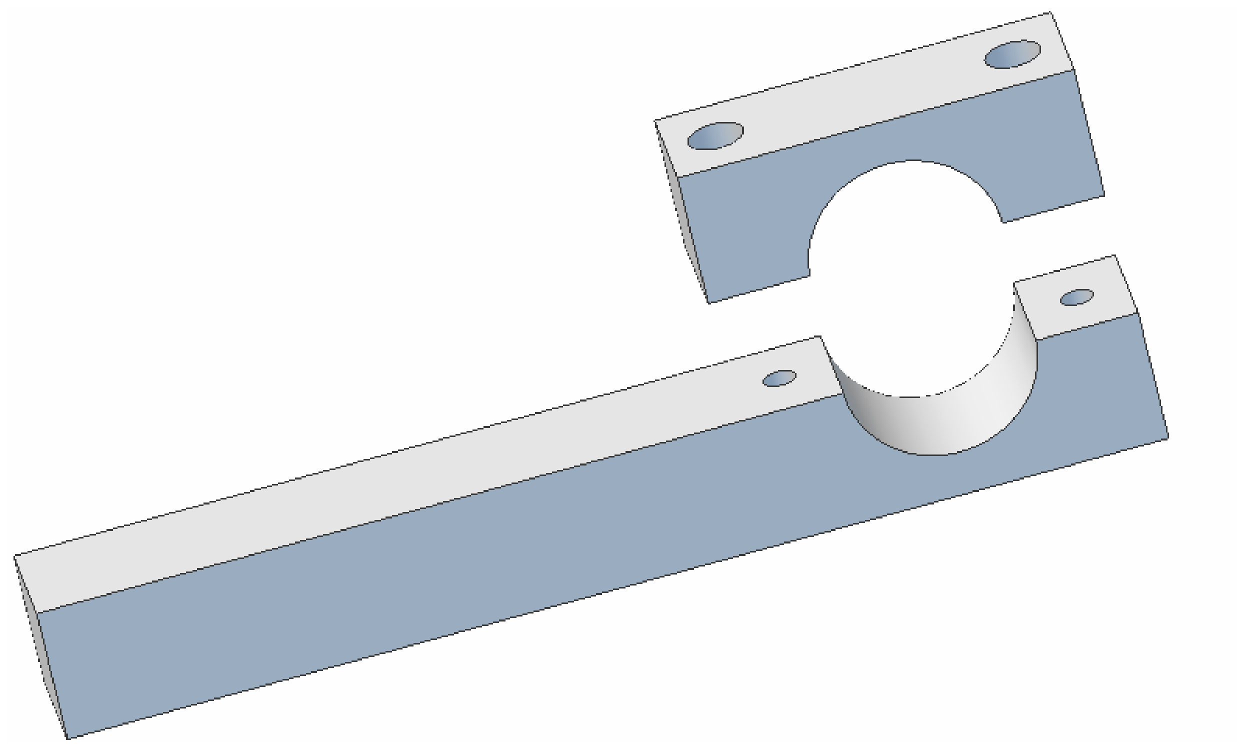
You might be able to see the approximately 0.014" gap between support bar and clamp bar. The High Speed Rotary Tool is securely held.

Just Like Downtown

The final step once I am confident that the project is worth drafting up, is to capture the finished design. I am using the Computer Aided Design tool called Alibre here.







R. G. Sparber

rgsparber@aol.com