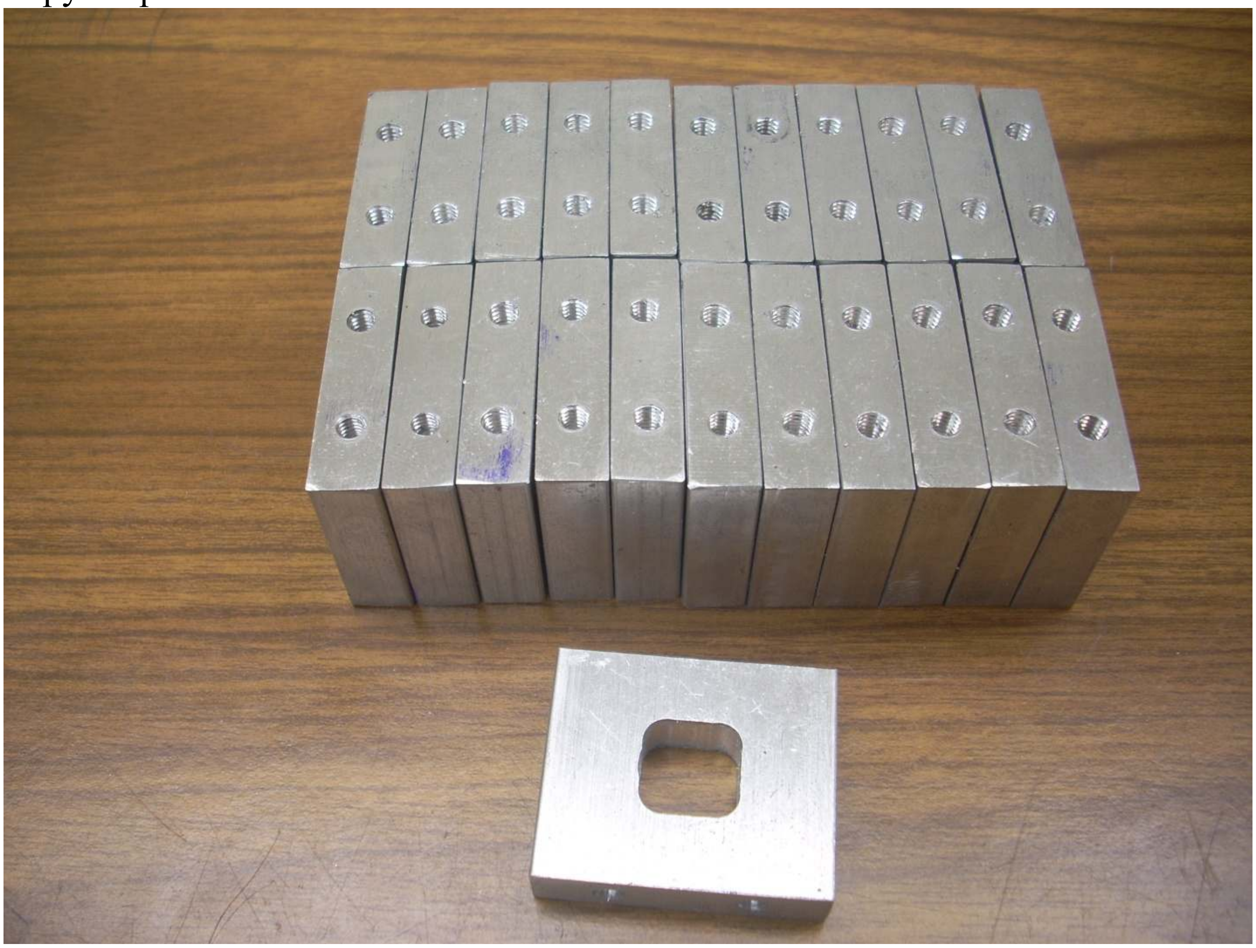
An Example of Small Scale Production

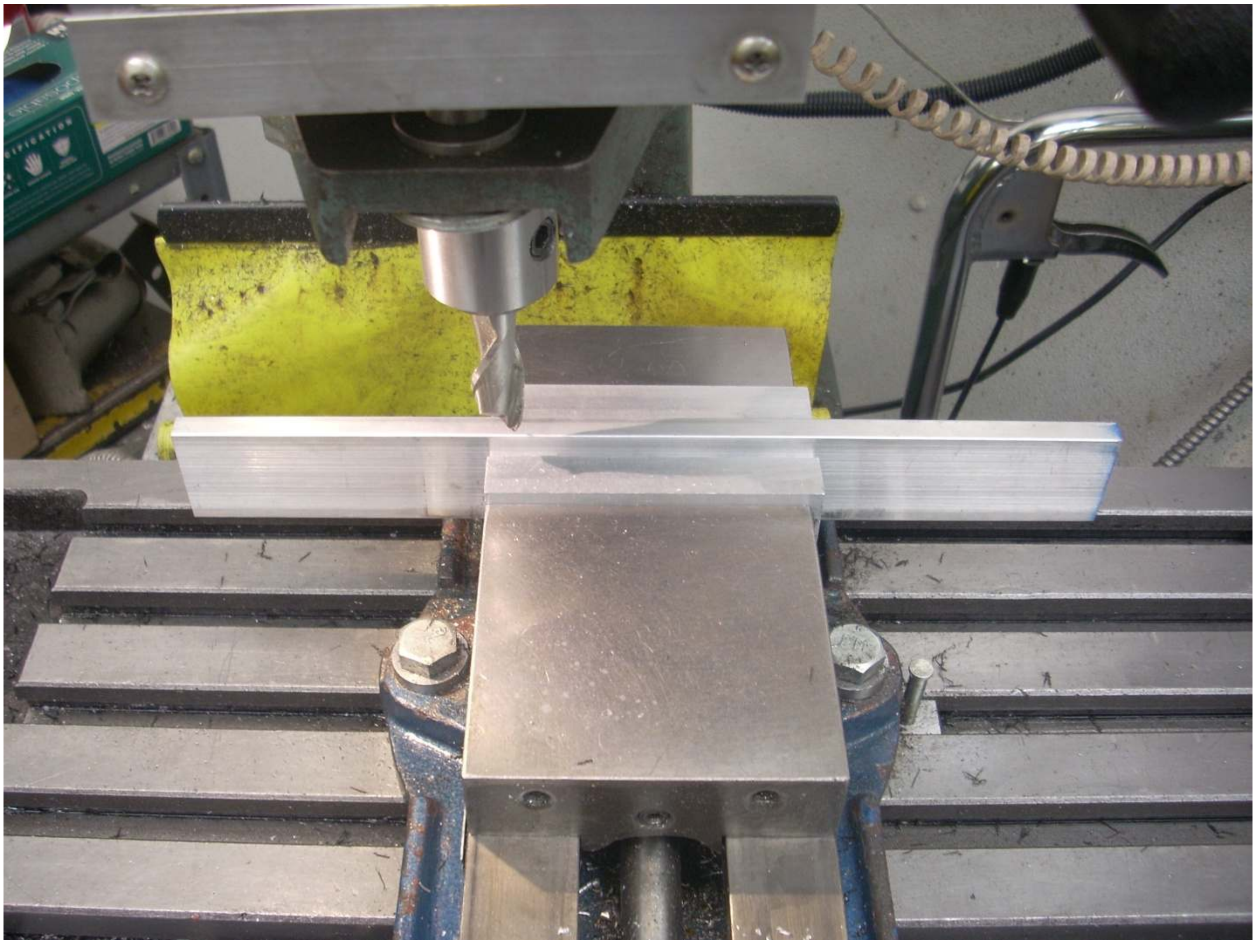
By R. G. Sparber

Copyleft protects this document.[1]



A friend of mine needed a small quantity of this part. If I had to make just one of these, I would approach the fabrication one way. But when more than a one part is needed, it is time to think about "production" methods. This article presents the method I chose.

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



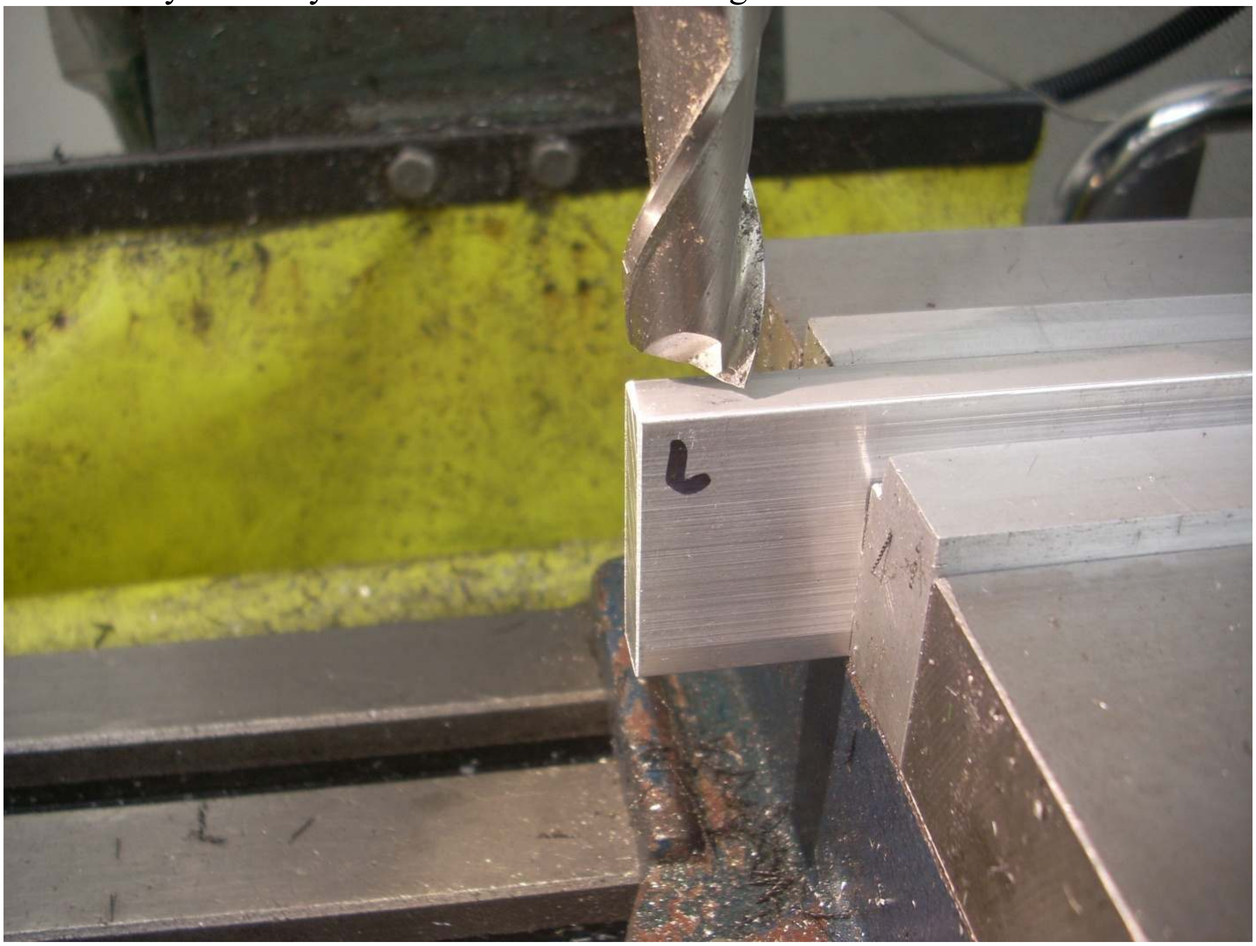
The parts must be 3/8" thick and 1.491 ± 0.005" wide. I was able to find 3/8" x 1.5" x 12" bar stock. My vise is 4" wide. So if I just clamped the bar stock in the vise to mill the width, there would be excessive overhang. So instead I cut each bar in half.

Most of the time in my shop, I am making "one-offs" that are cut to fit with other parts. But with this part, I must hit an absolute number. To do this, I used my spacer blocks to define a width of 1.491 ± 0.0004". This is a worst case error given that each block has an error of ± 0.0001".

Note that my caliper reads 1.493" so clearly the caliper is not as accurate as my stack of blocks. Furthermore, the caliper has a display repeatability of ± 0.0005". There is also error associated with my skill to hold the caliper square to the pieces. By repeated measurements, I can minimize this skill related error.

With the caliper on the blocks, I zeroed it. In this way it will show deviation from my reference rather than the absolute distance.

I took my first 6" long bar and measured the width change at each end. At the right end I set my zero. My left end showed a widening of 0.0035".

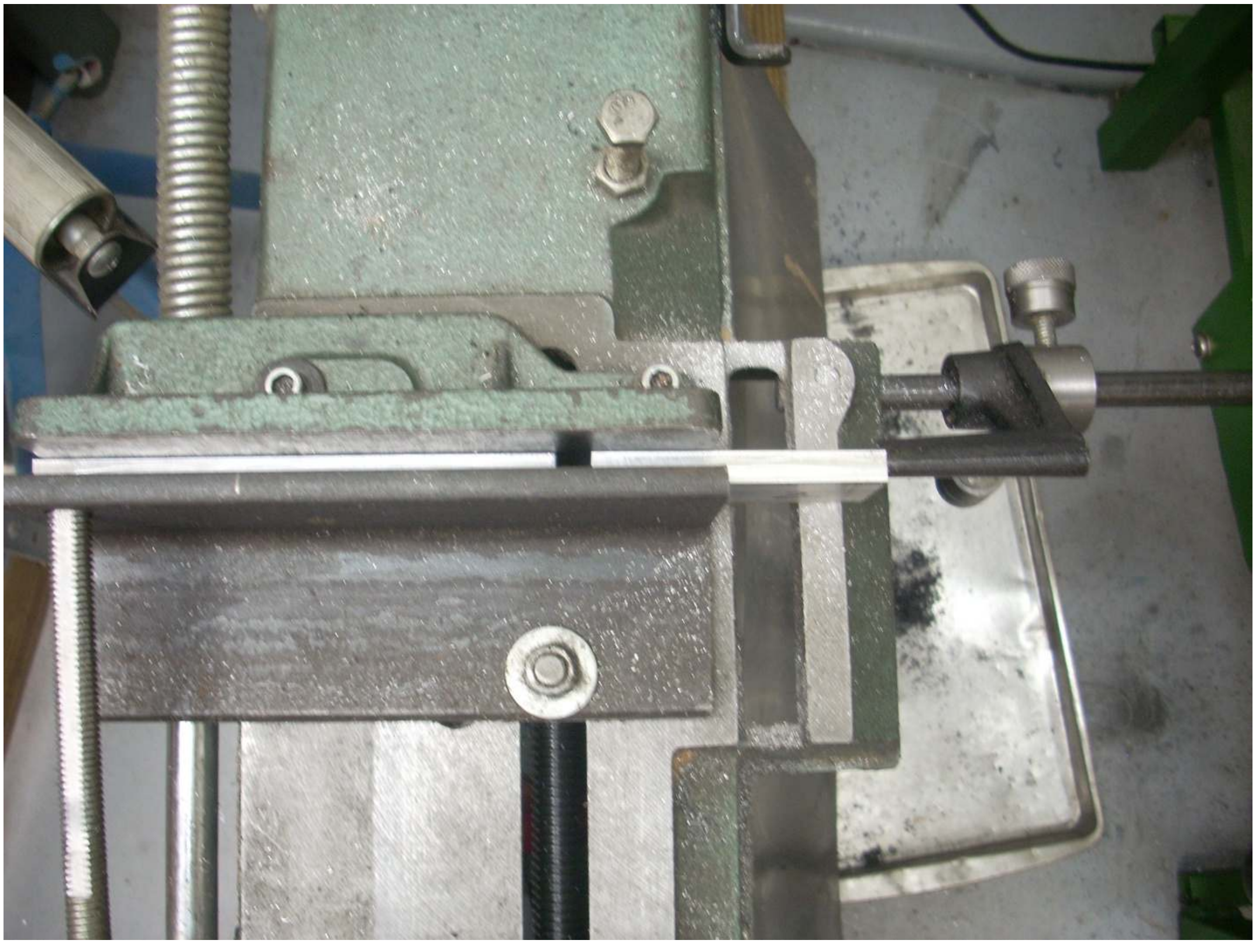


I then clamped the bar in my vise and brought my 5/8" two flute end mill down on the right end. My DRO was zeroed. Moving over to the left end, I brought my end mill down to just touch the surface. I read a rise of 0.0035". This tells me my part is squarely bedded in the vise.

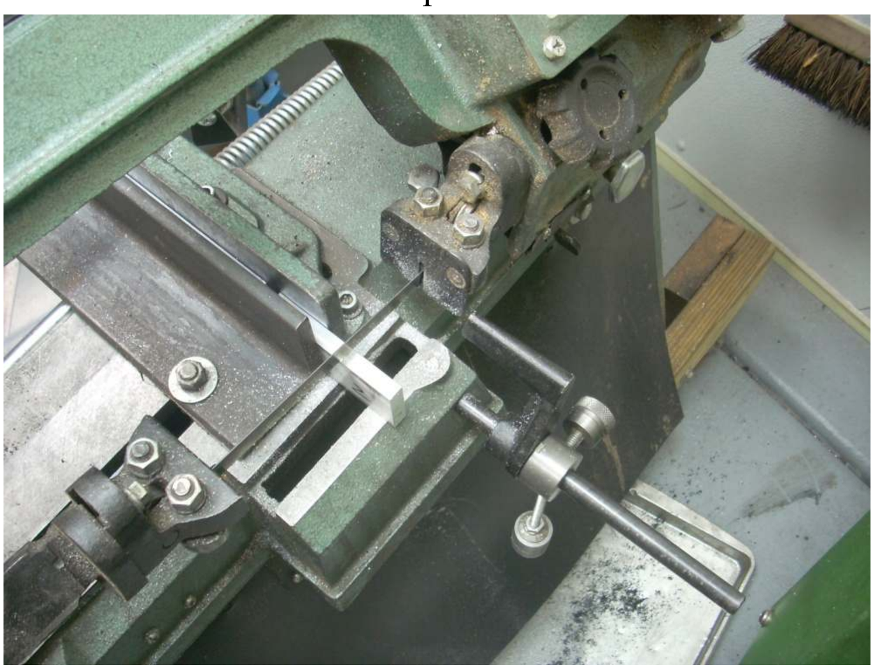
I then took a light cut and zeroed my Z axis. This establishes my initial reference surface. I then set my caliper to zero with it pressed against my stack of spacer blocks. The caliper was then used to measure how much more I must mill from the edge to get my finished width. When done, the bar was removed from the vise, deburred, and cleaned.

Within the repeatability of the caliper, I hit my number. Now, it might sound like I was being a bit "anal" setting the end mill height, but I used this height for milling all of my bars to width. So some care at set up gave me a good payback later. Even a small error in set up should keep me within my spec of ±0.005".

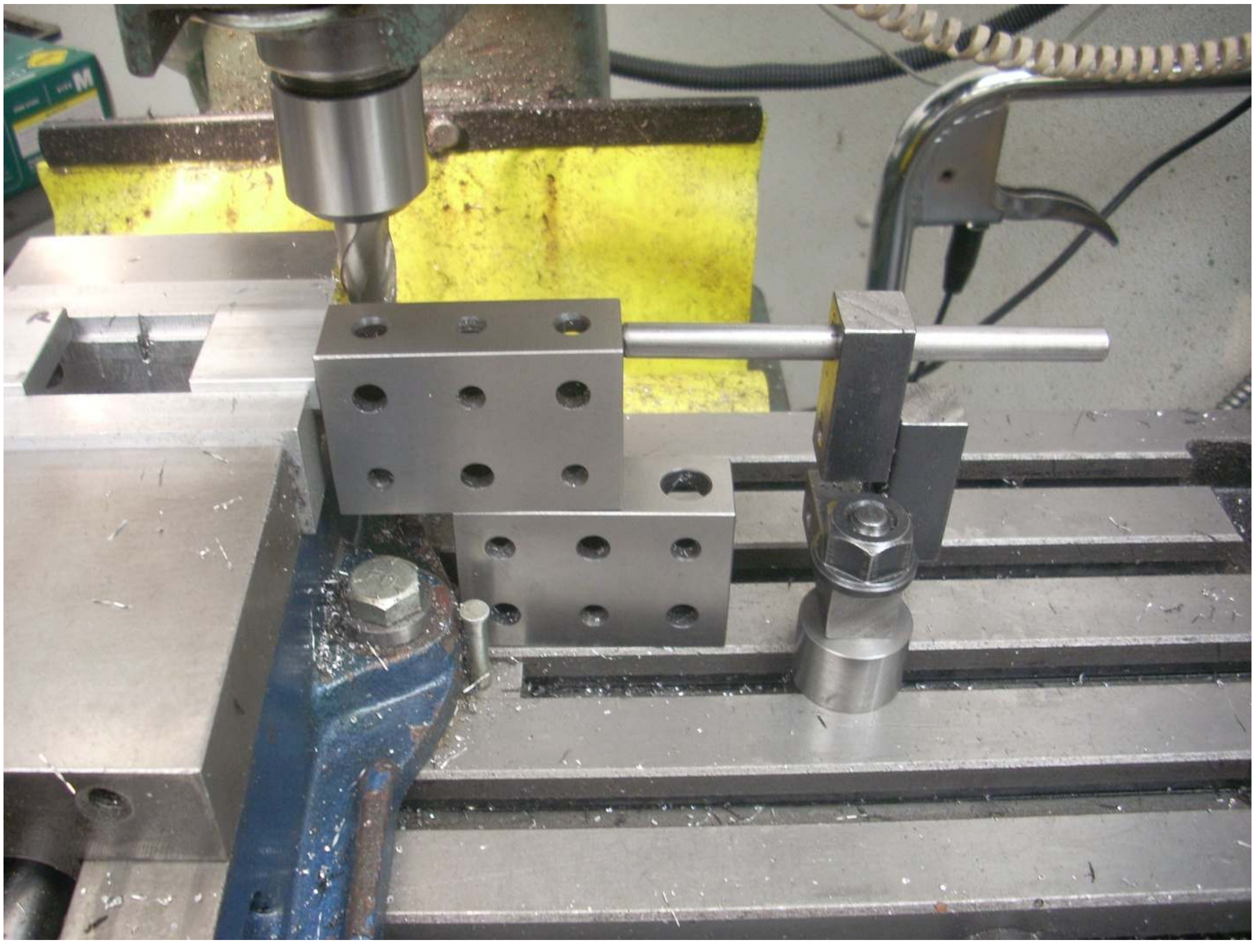
I then carefully cleaned the vise, bed down the next bar, and milled it to width. The cycle was repeated for all 6 bars. A check of the finished bars showed all were within 0.001" of ideal except for one that was 0.007" too narrow at one end. My guess is that this bar had a narrow point in it. Not a problem, I made a few extra parts. In the end, I lost only one part to this defect.



The next step was to cut each 6" bar into 1.3" long blocks. This will give me 0.05"of material to square it up on my mill. The first block was carefully set in my bandsaw vise and the stop set.



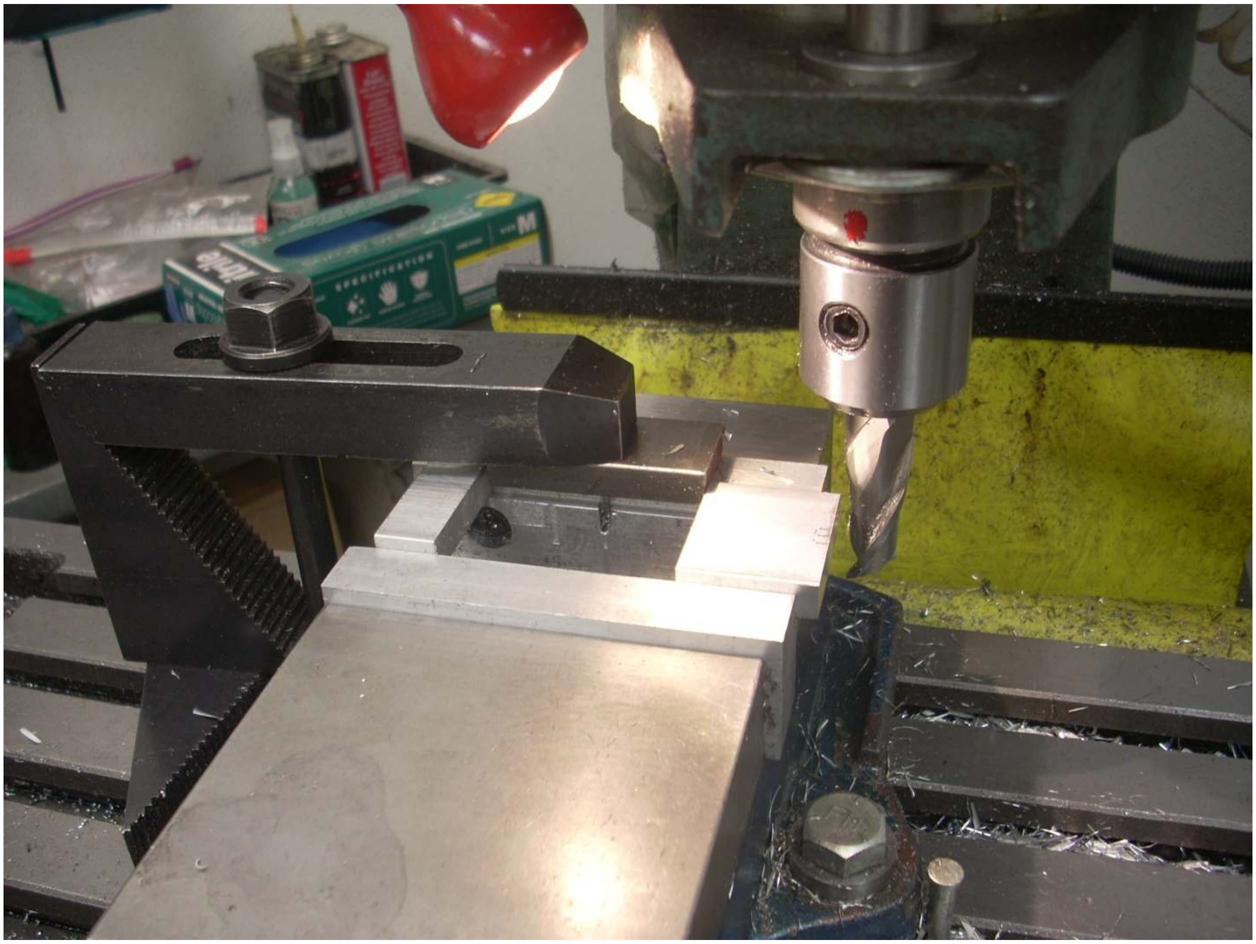
Before starting the saw, I moved the stop out of the way. Failure to do this can cause the part to twist and jam as the blade finishes its cut.



The next step was to square up one edge of each block. I could have put up a stop that touched the left edge of the block but did not trust that my saw cuts were all that square. So instead I set up a stop on the edge about to be cut. The two 1-2-3 blocks let me position the work piece.

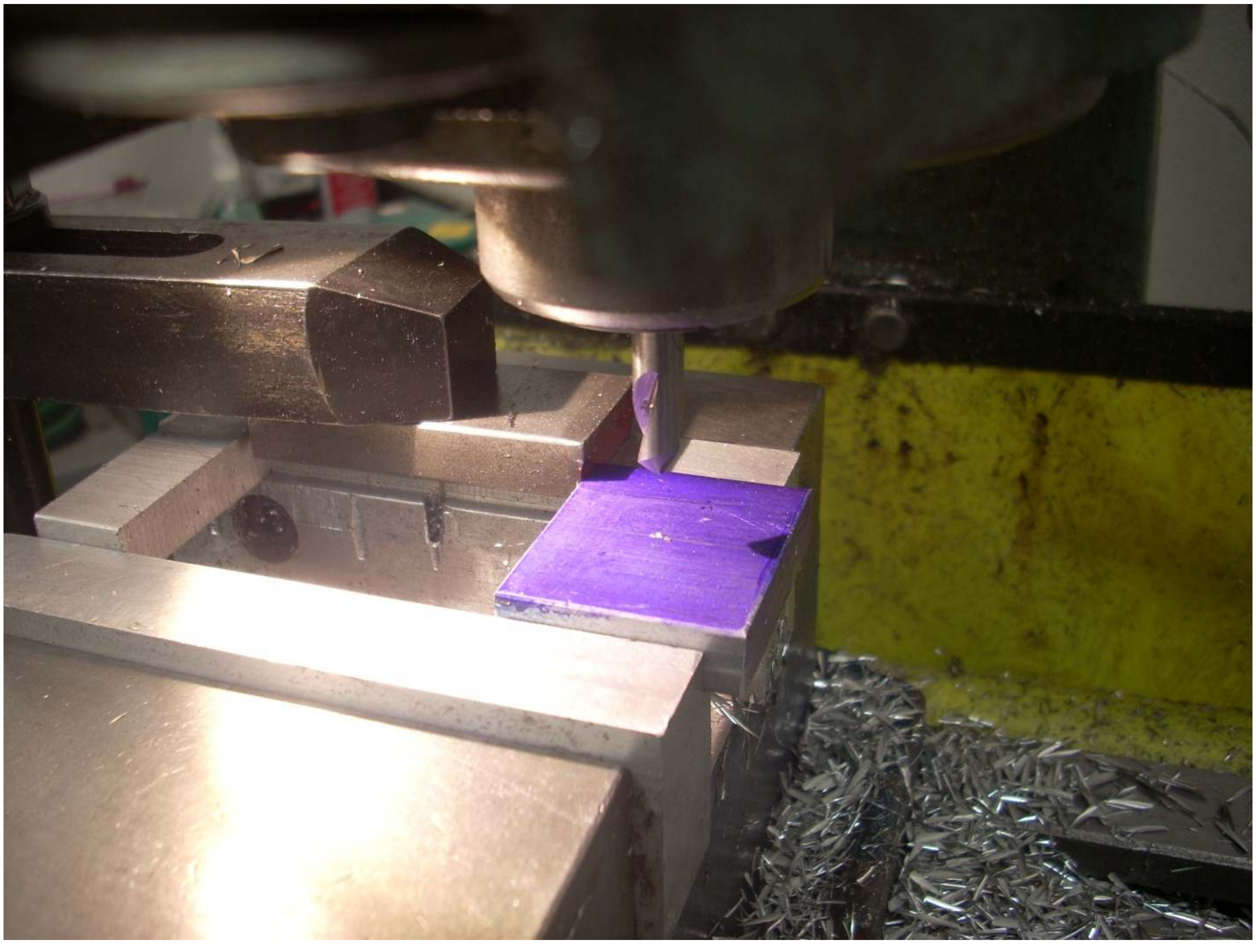
Once clamped, the 1-2-3 blocks are removed. I then took a light cut to define my reference surface. Then I fed in another 0.01" to be sure I always cut the full width.

I then had a stack of blocks of known width and one end square.



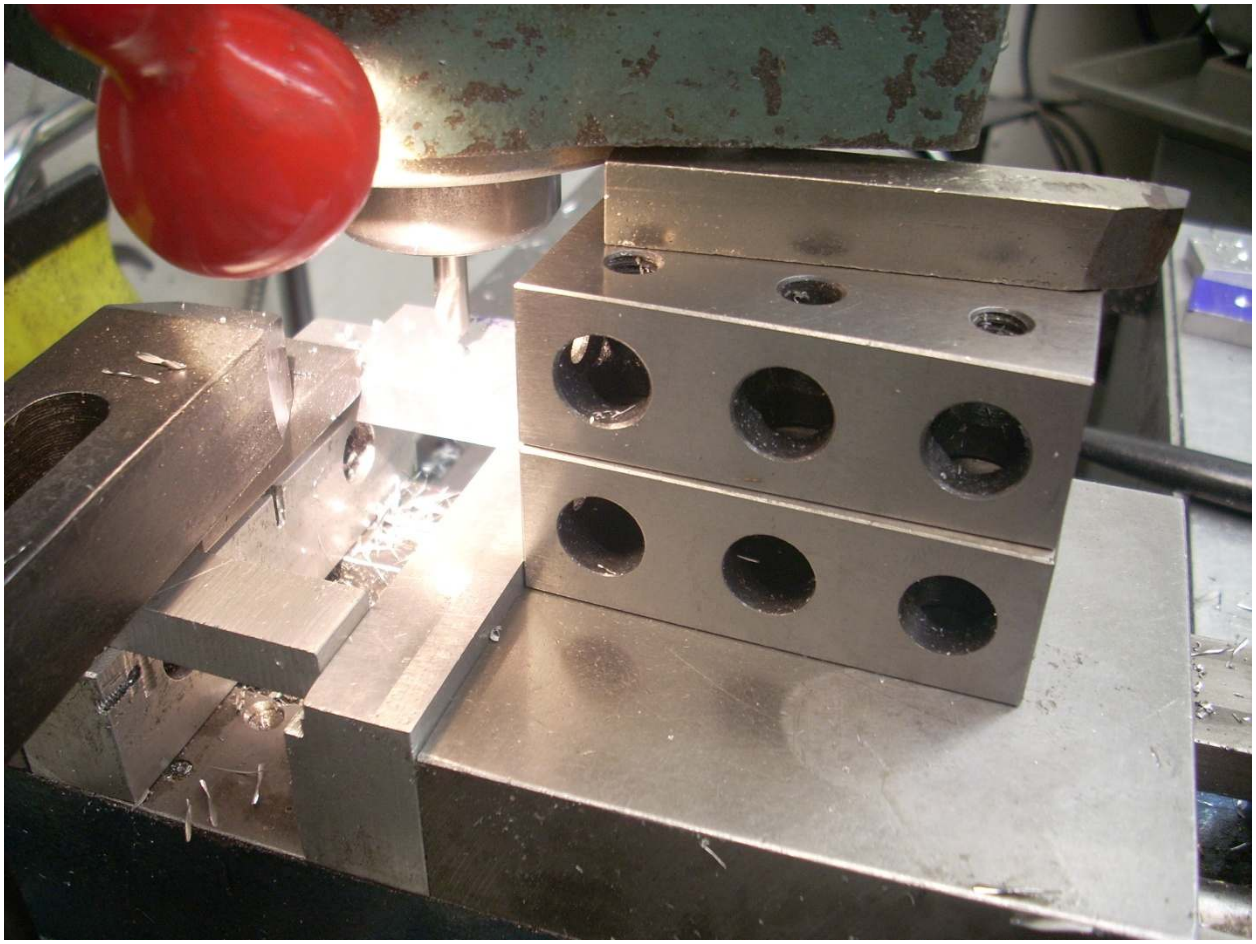
Next I set up a fence on the left edge. This edge was the one that I squared up in the last step. I took a light cut, measured the part's width, and adjusted the X axis to cut the finished dimension. The first part was checked and a fine adjustment was made. The DRO's X axis was set to zero with allowance for the radius of the tool. Then I was able to cut all of the blocks with no further changes to the mill. I did do spot checks of the cut parts to be sure nothing shifted.

So far, so good. I have my one block that is too narrow but the rest came out identical. This uniformity is essential even on dimensions that are not critical. Without it, my stops may not give me uniform feature positioning.



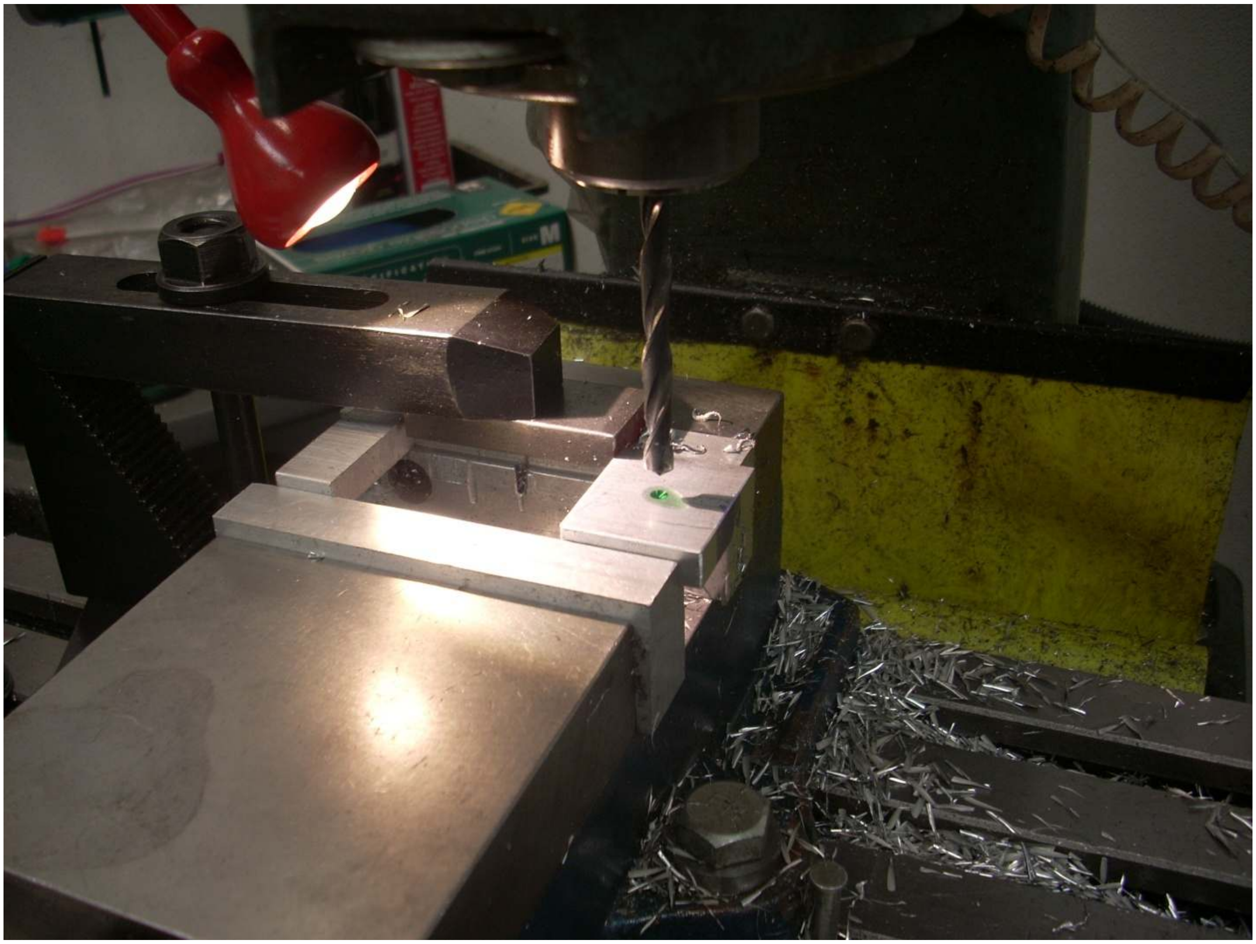
My DRO's X axis was set to zero while cutting the right edge. I used my Electronic Edge Finder to set the Y axis zero. From there I was able to dial in the center of my square hole. This became my new (0,0) position. The plan was to knock a 1/2" diameter hole and then use my 1/4" end mill to square it up.

Here you see the mill set up with a 1/4" spot drill held in my 1/4" R8 collet. I had previously trial fitted holding my 1/2" drill in a 1/2" R8 collet to be sure it fit. I didn't want to have to move my mill head once this process started.

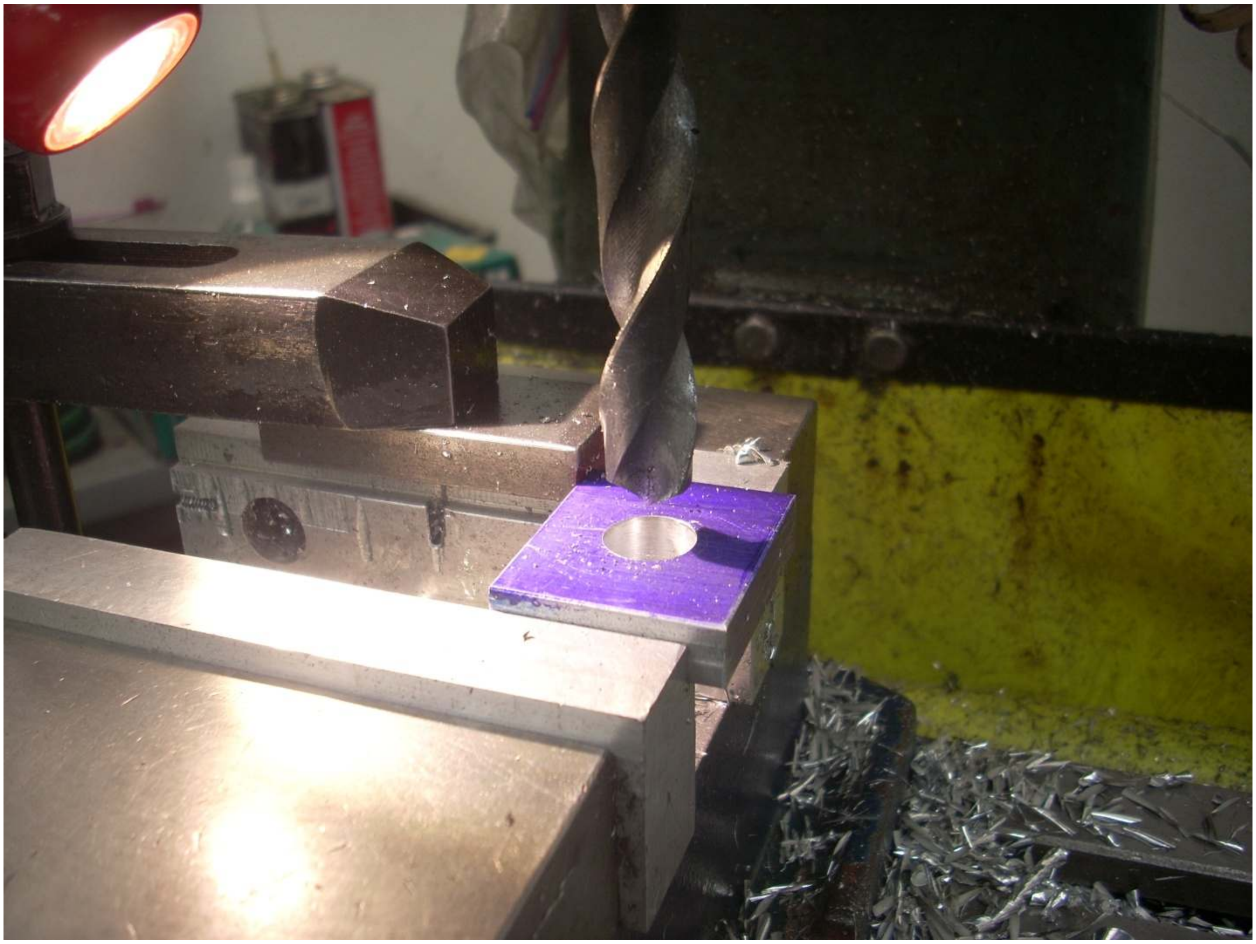


Drilling the cone shaped hole with the spot drill is not a critical step but that doesn't mean that I could not benefit from using a stop. I put down my two 1-2-3 blocks with a cutter on top. In this way I just drop in the part, pull down until the bottom of the spindle bearing housing hits the stop, and back the cutter away. Quick yet repeatable.

In no time I had all blocks drilled with the spot drill.



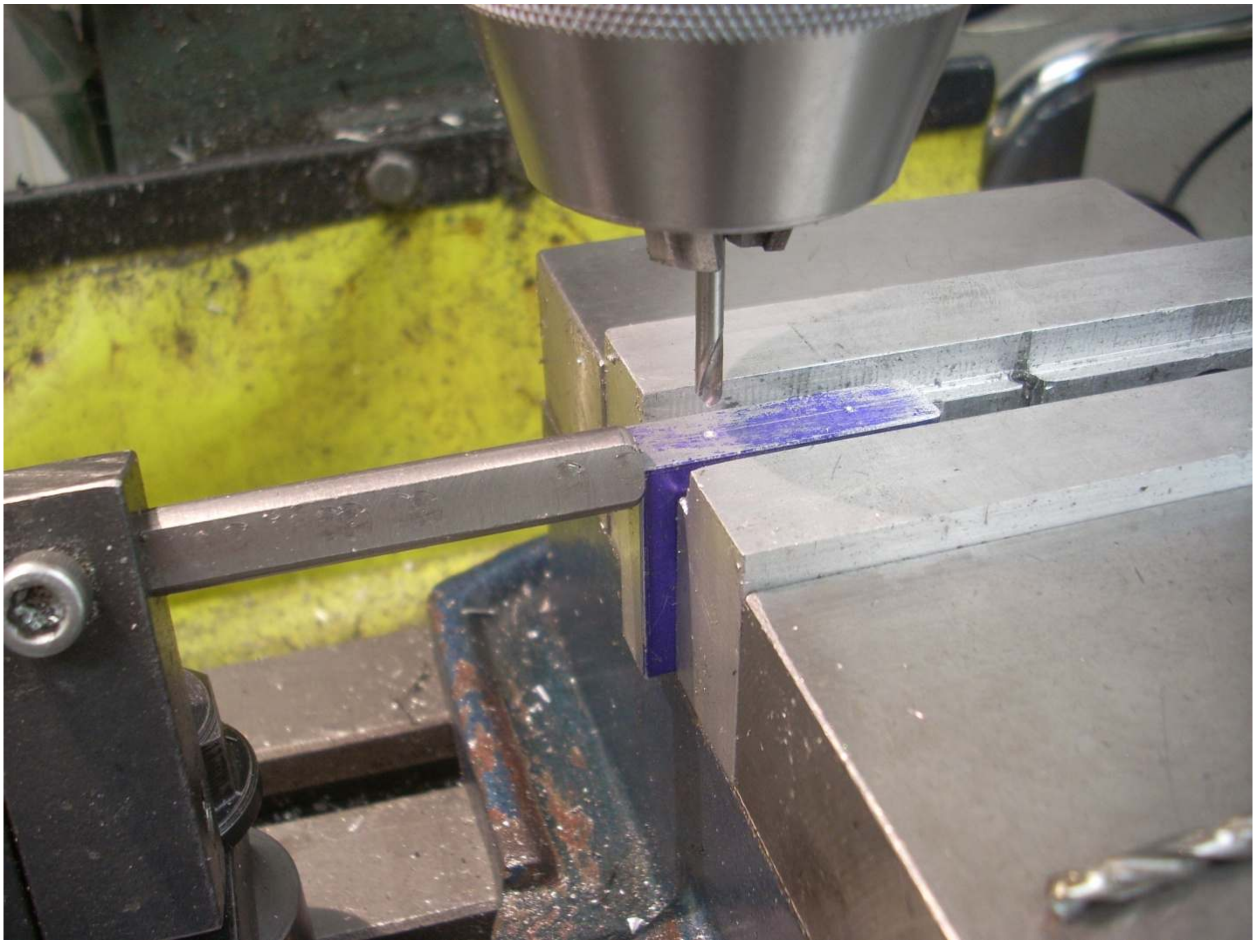
The next step was to install my 1/4" drill and put a tiny drop of cutting fluid in the cone shaped hole left by the spot drill. I am through drilling now so it is easy to just feel for the drop in resistance as I broke though the bottom of the part. Again, all blocks were drilled. I didn't bother to deburr because the reference surfaces held in my vise were still smooth and clean.


With the 1/4" R8 collet replaced by my 1/2" collet, I mounted my 1/2" drill. All blocks were drilled through.

The final milling step for the square hole requires my 1/4" end mill. I swapped out my 1/2" collet for my 1/4" collet and locked in this end mill. Because I used collets for all of these operations, I didn't have to raise or lower my mill head which would have required me to reestablish my zero.

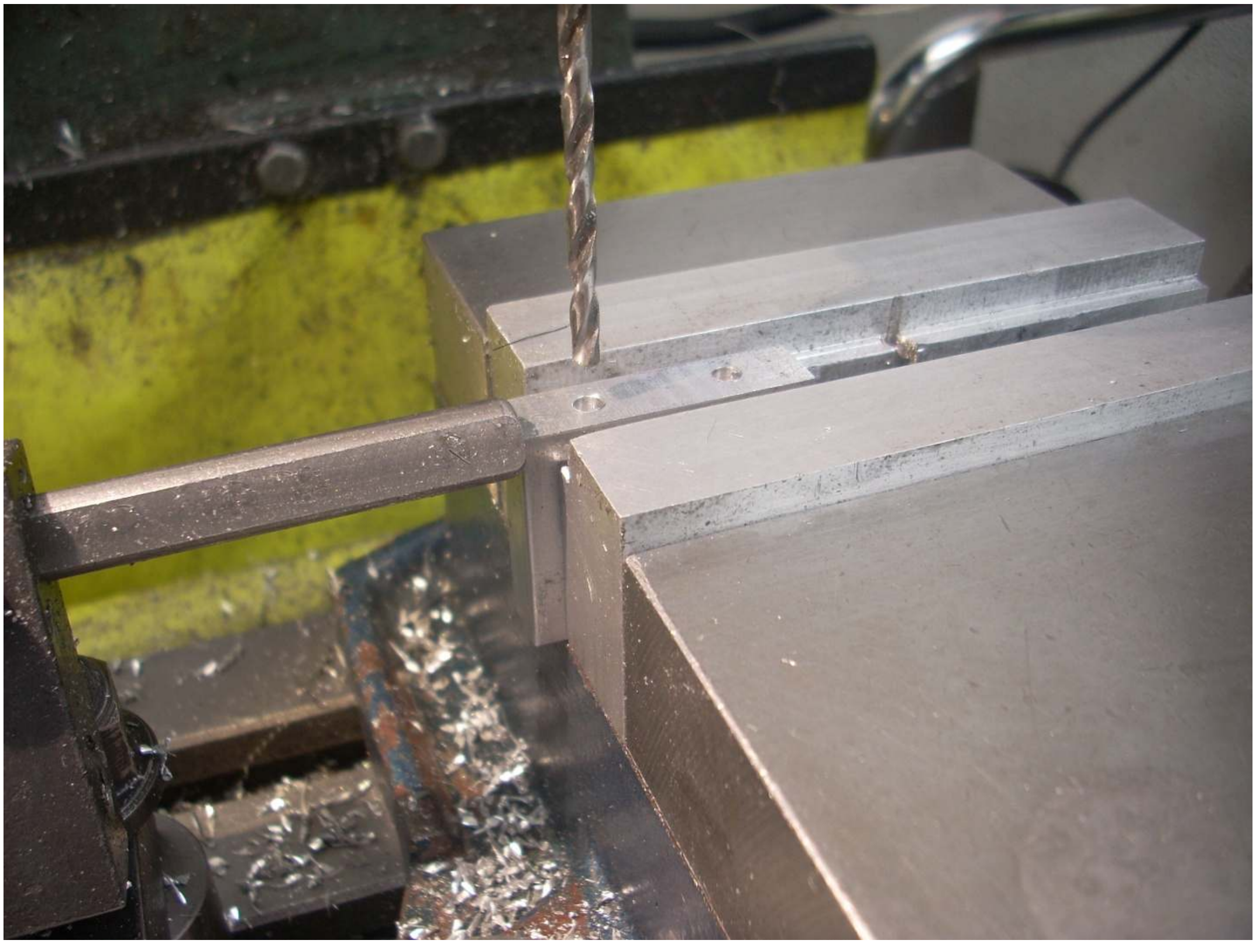
I did not fool with tool offsets on my DRO to cut this square hole. Instead I defined a tool path for the center of the end mill. In this way I don't have to keep changing the tool offset to match the face being cut. Furthermore, since the hole is square, I only had to remember one number for all cuts. I moved the X axis to +0.131, locked the X axis, unlocked the Y axis and moved to +0.131, then locked the Y axis. The X axis was again unlocked and I moved to -0.131. The pattern continued until all four inside surfaces were cut. This approach minimizes the chance that I would get distracted or confused and go too far.

Notice that this part does have layout lines on it. My first part was done this way to avoid making BIG mistakes. The rest of the parts had a dab of layout fluid on them to mark which end was my reference. The square hole is closer to this reference end than to the opposite end.



Cutting the square holes was certainly demanding. I got to relax with the next step. Using my 1/8" diameter spot drill, I drilled the location of the two end holes which will later be tapped.

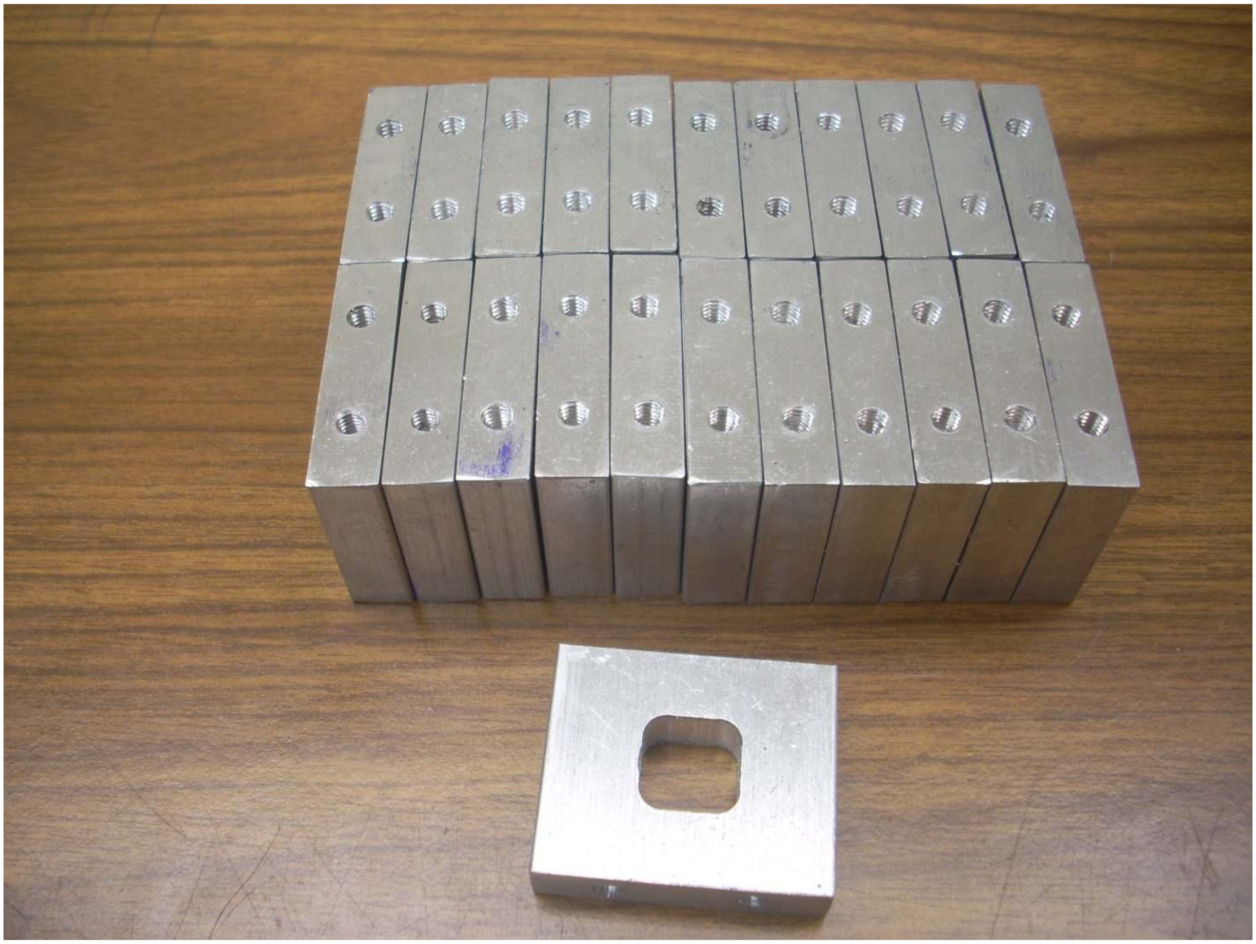
The holes are on the center line of the part and 0.390" from the edge. Because the width of the part was held so accurately, I can put my stop on one edge, drill, flip the part around, and use my stop on the other edge. Otherwise, I would have to drill the first hole, crank over 1.101", and drill the second hole.



Without disturbing the fixture, I ran my #29 drill down about 3/8" for all blocks.

This last step was, by far, the most fun. I have a Tapmatic® tapping head that is simply not worth the trouble for one hole. But when faced with 46 holes to tap, it is great.

I had to raise the mill head due to the tapping head's additional length but that was not a problem. The head compensates for small alignment errors. I did keep the stop on my vise. Note the vertical bar behind my stop. It stops the tapping head bar from rotating. All I had to do was feed down and a split second later, pull back up to reveal a perfectly tapped hole. The fact that the hole was blind posed no problem at all.



Job done.

I welcome your comments and questions.

Rick Sparber
Rgsparber@aol.com
Rick.Sparber.org