Machining and Fitting the Clapper Box

by R. G. Sparber

08/17/2008

It is really nice to get a second chance. There were a few things I did wrong when machining the down feed casting but was certainly not about to make another casting just to try again. The clapper box is similar to the down feed so this time I get to apply my new found skill and hopefully not make the same mistakes twice.

Preparing The Rough Casting

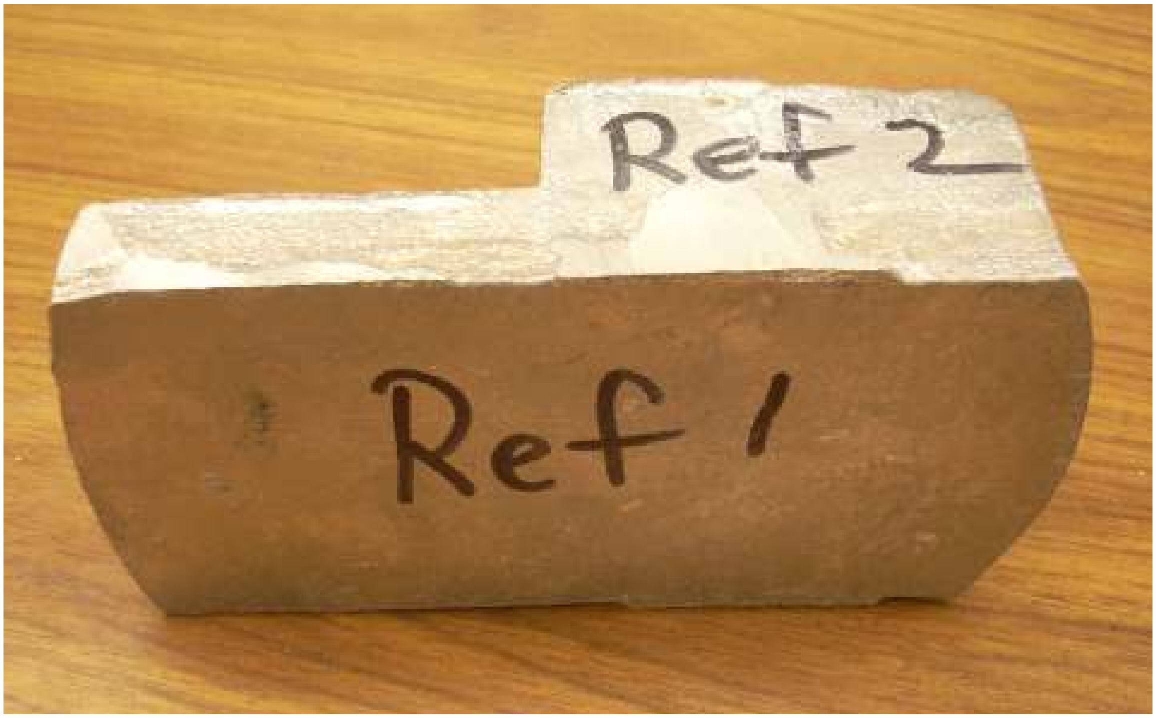
This awful looking casting is the last one from my worn out Petrobond. As bad as it looks here, it sure did machine up nicely.

The back side looks better.

A few minutes of sawing removed the gate system and bits of flash.

I know, I know, it still looks ugly!

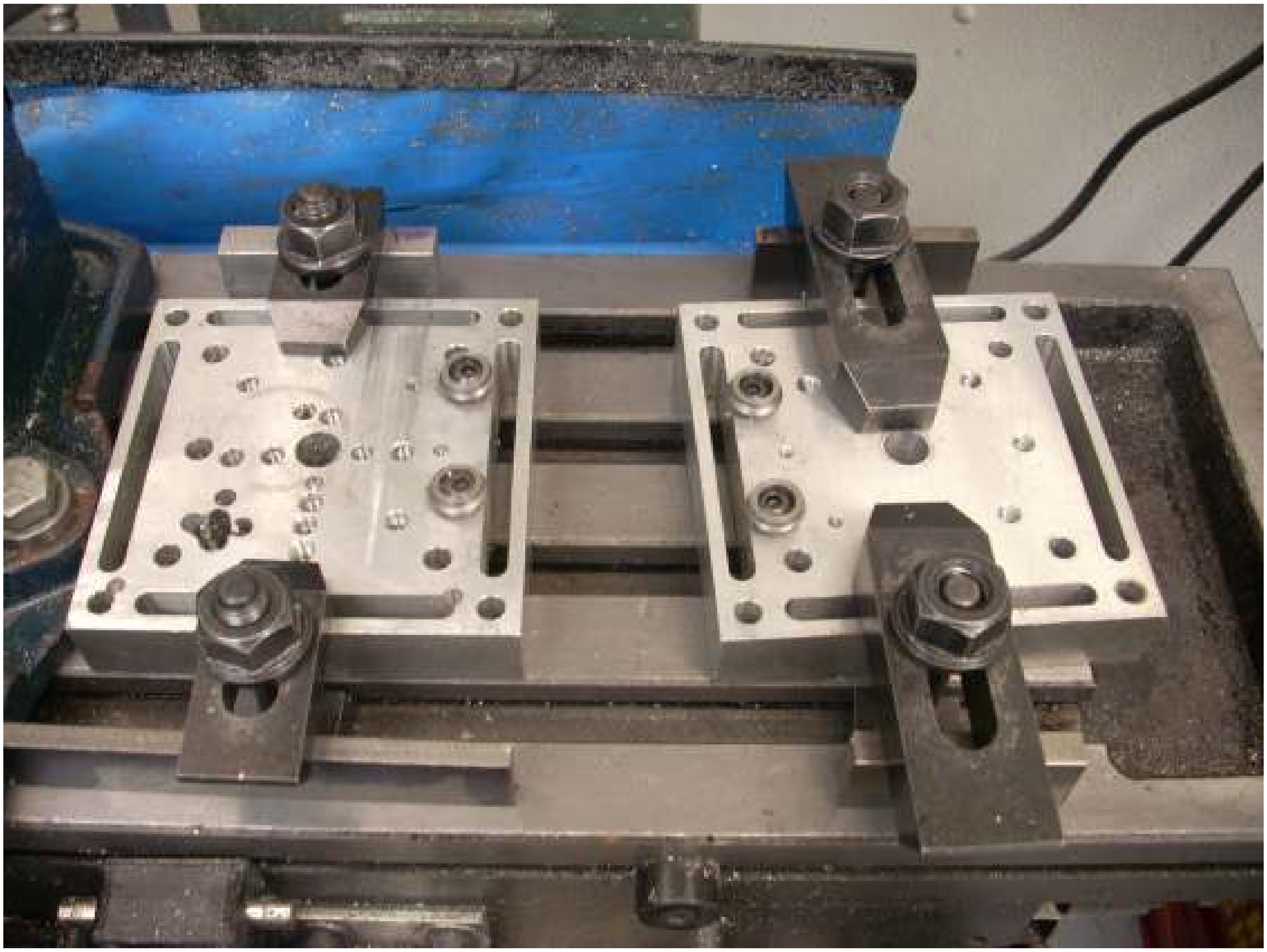
Defining the Machining References



The first step is to define the reference surfaces. Reference surface 3 is the step at the midpoint of the casting. Once these references have been cut, all other features will be defined relative to them. This makes locating features much easier plus gives better accuracy than trying to find out where you are before each operation.

Parallel to “Ref 1” is the top face of the casting. Ideally it is perfectly parallel but in reality, nothing is perfect so I need to give each a unique name. To minimize confusion, I will call “Ref 1” as marked above “Reference 1 primary” and the top face “Reference 1 secondary”. Similarly I have a Reference 2 primary and Reference 2 secondary. There is only one Ref 3 so it will be Reference 3 primary.

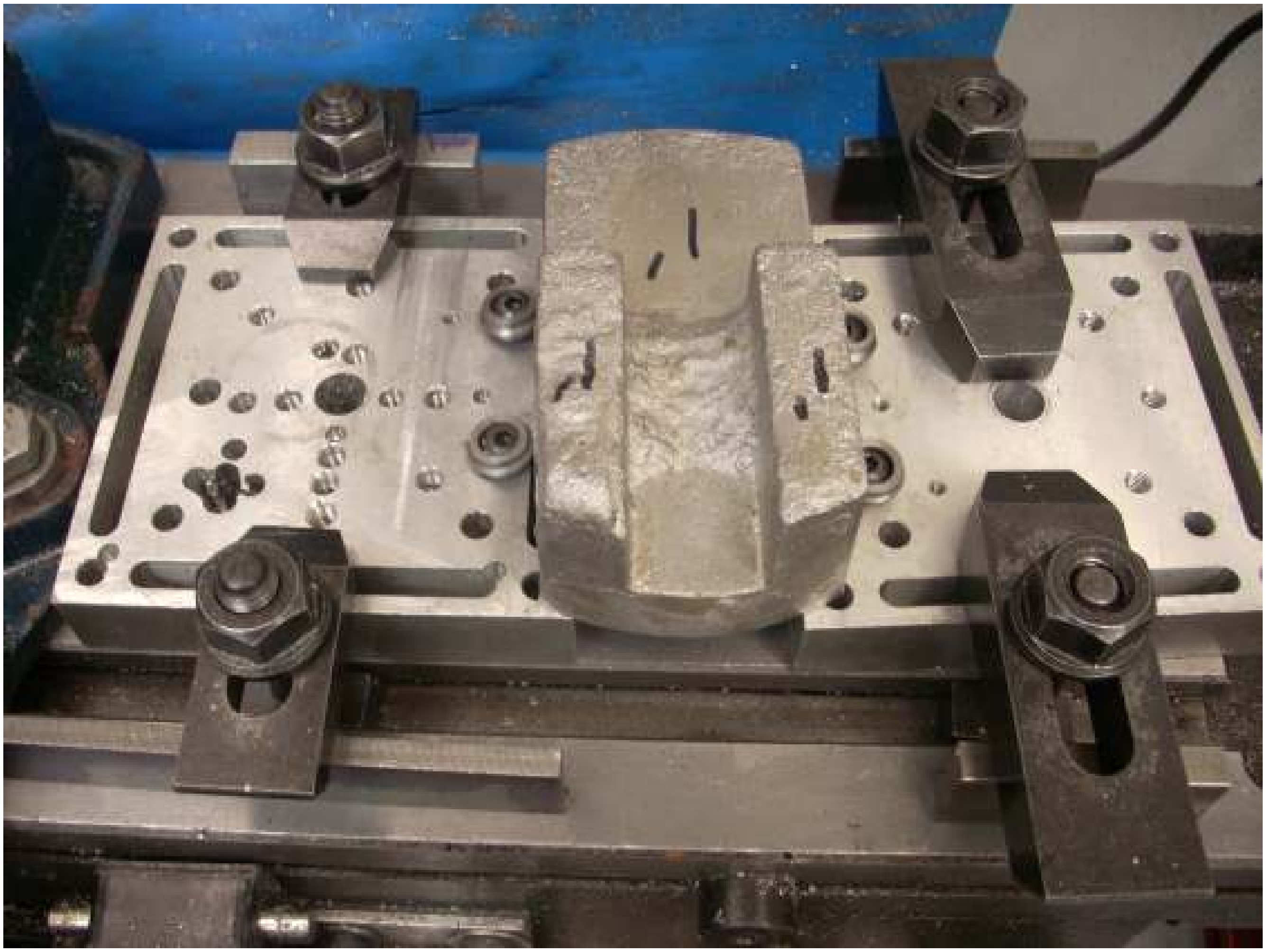
Holding the Rough Casting on the Mill/Drill



Now on to avoiding my first mistake from the last casting I machined.

I had used eccentric screws on one side of the casting and the vertical faces hold downs for the other side. The result was that the casting rose up about 0.005" on the hold down side.

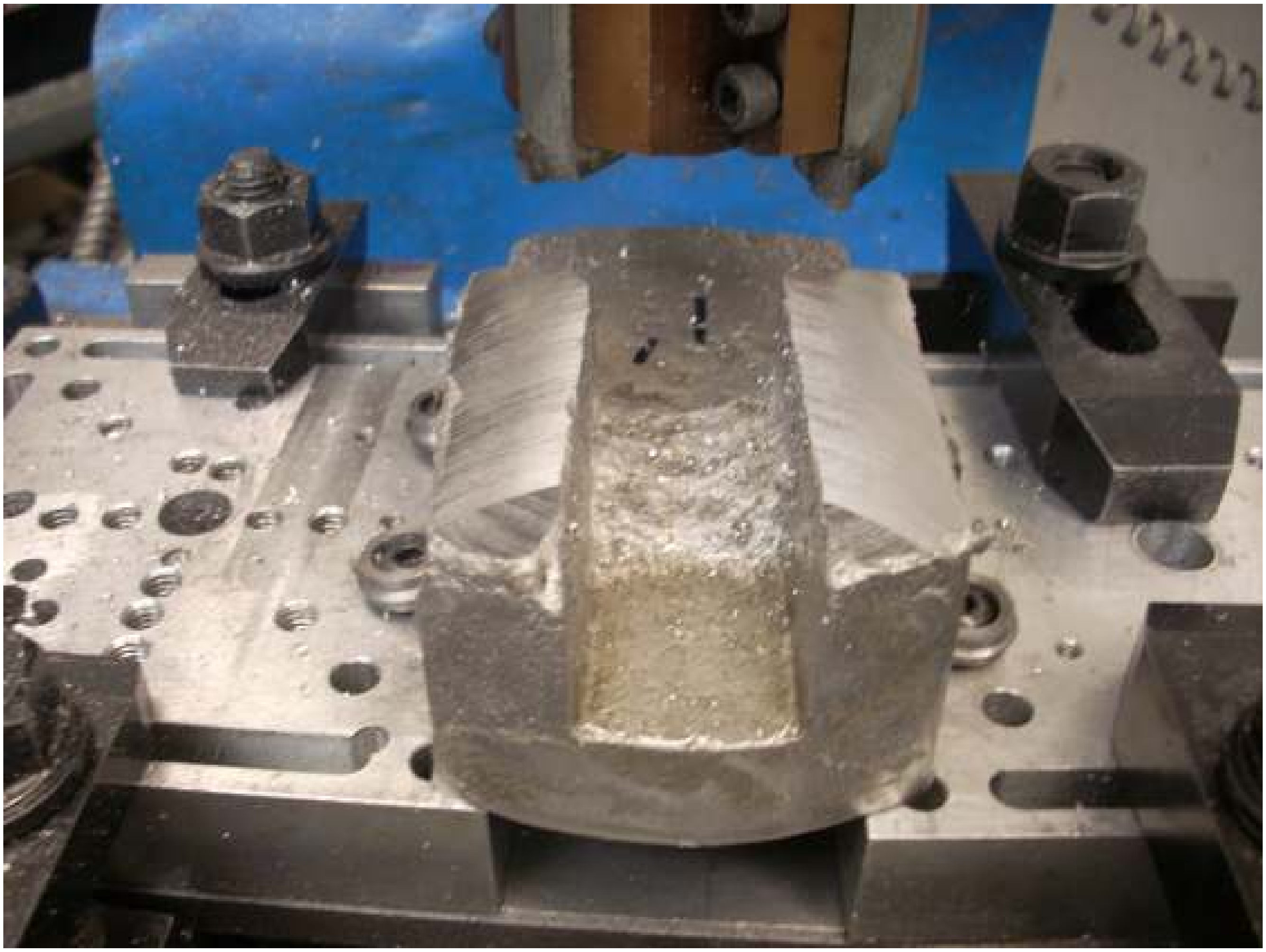
This time I will use eccentric screws on *both* sides. I start by turning all screws so the thin side faces the casting. The left plate was set roughly true to the mill table. The right plate was adjusted to be a close fit to the casting. Here I have removed the casting so you can see the set up.



By turning the eccentrics I was able to draw the casting tight against the support plates. My “Reference 1 secondary” is now ready for machining. Once it is cleaned up I will flip the casting over and cut “Reference 1 primary”.

Because I try to leave my vise bolted to the mill table, the support plate arrangement is off to the far right. I have rotated the head of my mill/drill so it can easily reach the casting. Try this trick with a square column mill/drill. See? Not all things are better with a square column mill/drill.

Cutting Reference 1



My Reference 1 secondary is now cut. It will be used to solidly support the casting as I cut Reference 1 primary.

Those small triangles at the forward end of each surface are due to my mill head being slightly out of tram. As you will see later, this has almost no effect on the accuracy of this cut.

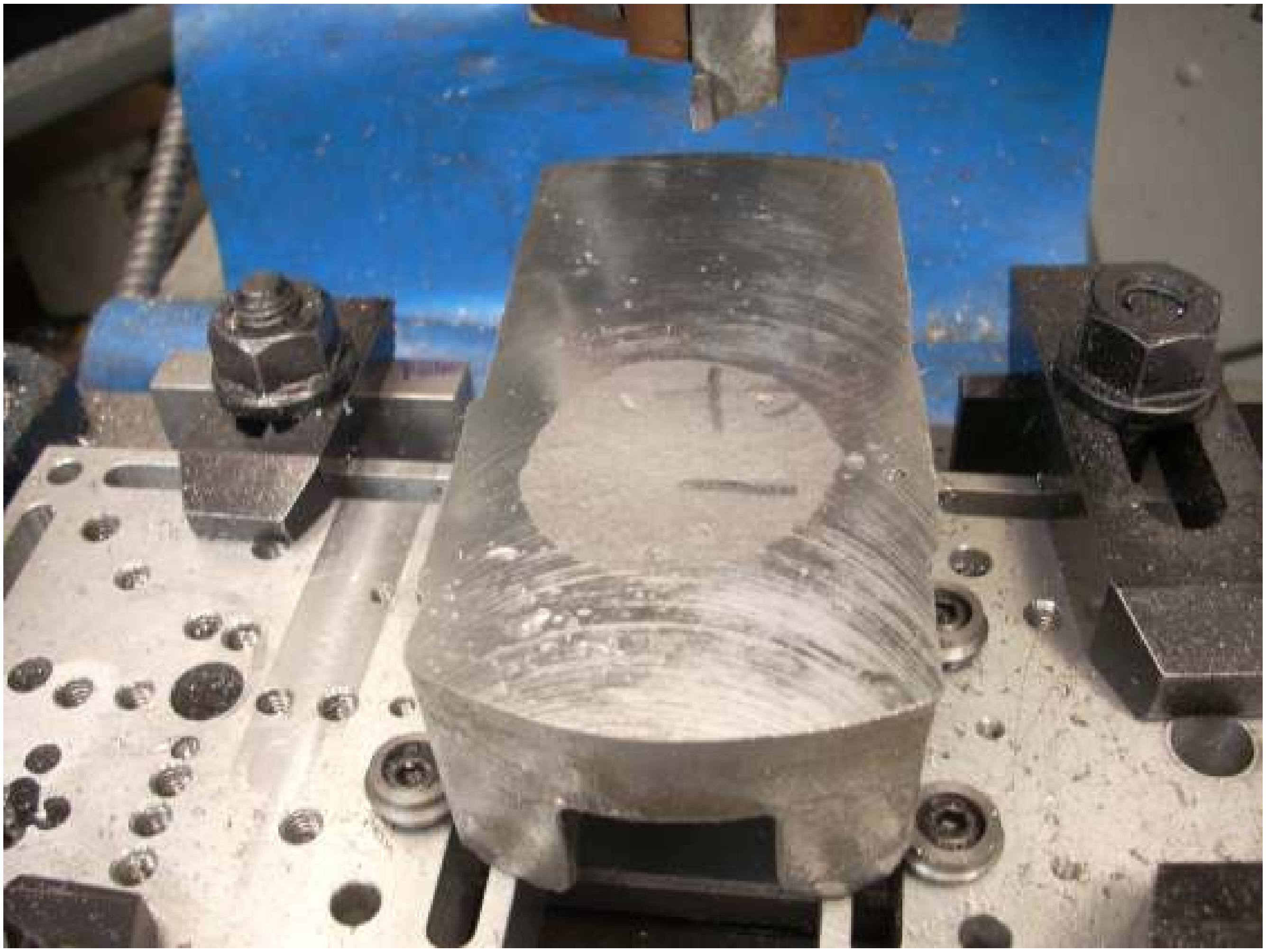
I can now turn the casting over and cut my Reference 1 primary plane.

Not shown is the effort taken to insure that all contact surface are absolutely clean. Lots of WD-40 and toilet paper[1] are used to remove every last speck of swarf.

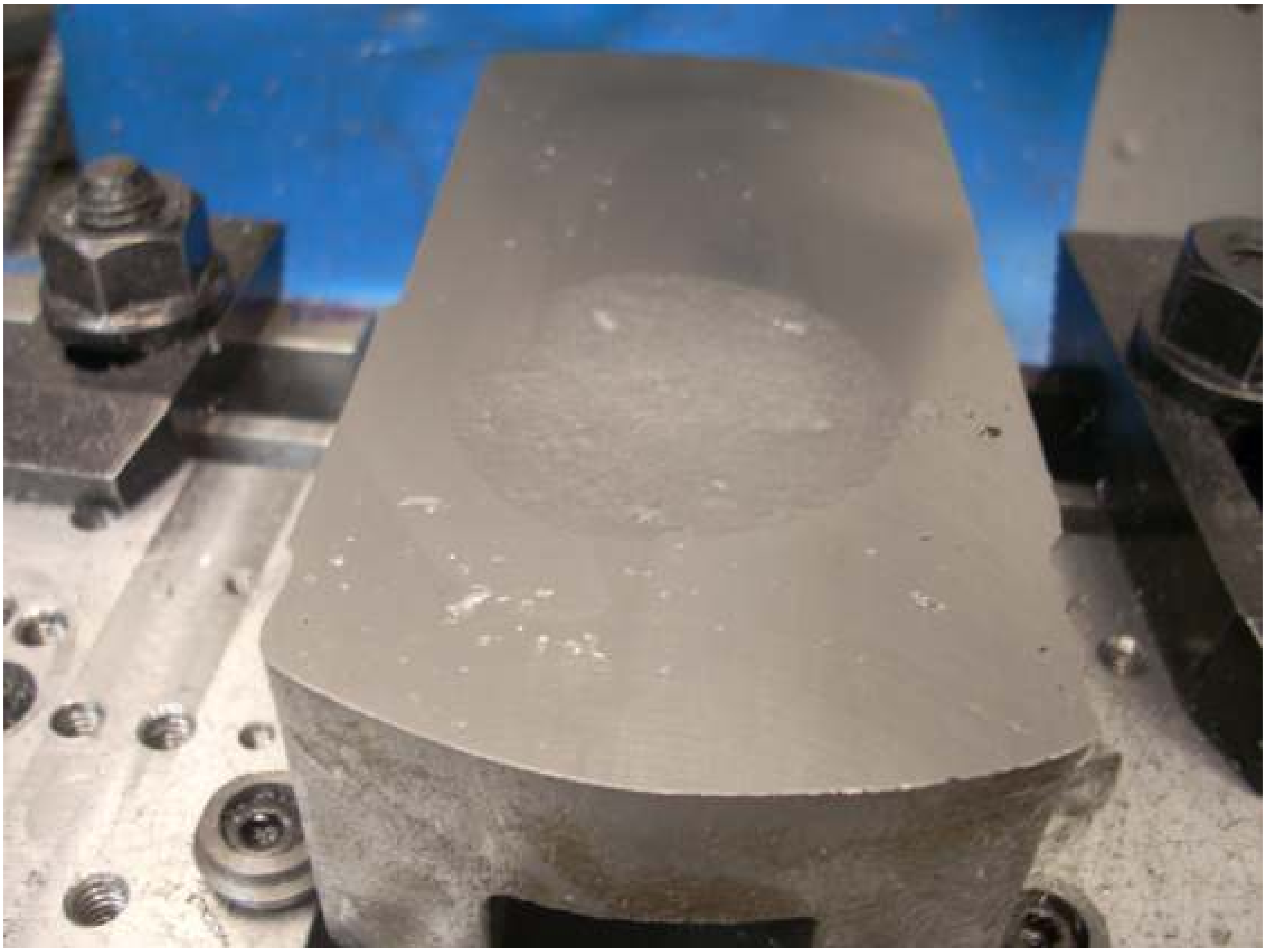
1 I steal Quilted Northern® from the house and it works fine.

The right block was unclamped from the table and repositioned to be a close fit on the casting. The eccentric screws are again tightened and Reference 1 secondary is snug down on the support plates. The top surface of the casting is now completely exposed and ready for a clean up cut.

After the first pass you can clearly see a bit of shrinkage. Fortunately I have plenty of metal here so can afford to cut deeper.



I took another 0.050” and the shrink area is now acceptably small.

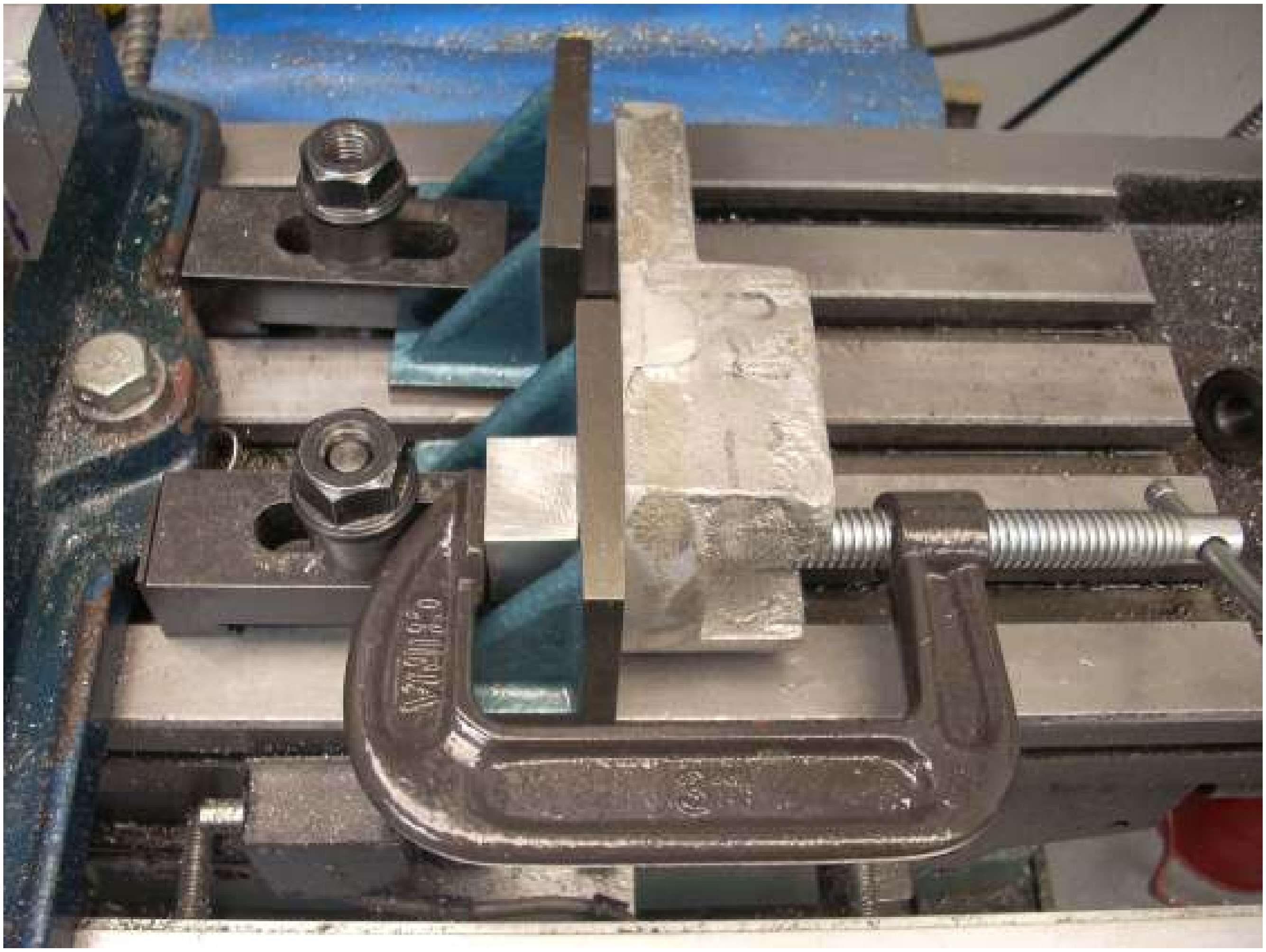

A few passes with a 3M pad and most of those nasty looking swirl marks are gone. This says they are very shallow. I now have a nice true Reference 1 primary.

To recap, I first put the uncut and rough Reference 1 primary down on the support blocks. It's opposite face, Reference 1 secondary was then cut. Reference 1 secondary is now flat.

Next Reference 1 secondary is placed down on the support blocks as you can see above and Reference 1 primary is cut. I now have a nice flat Reference 1 primary surface.

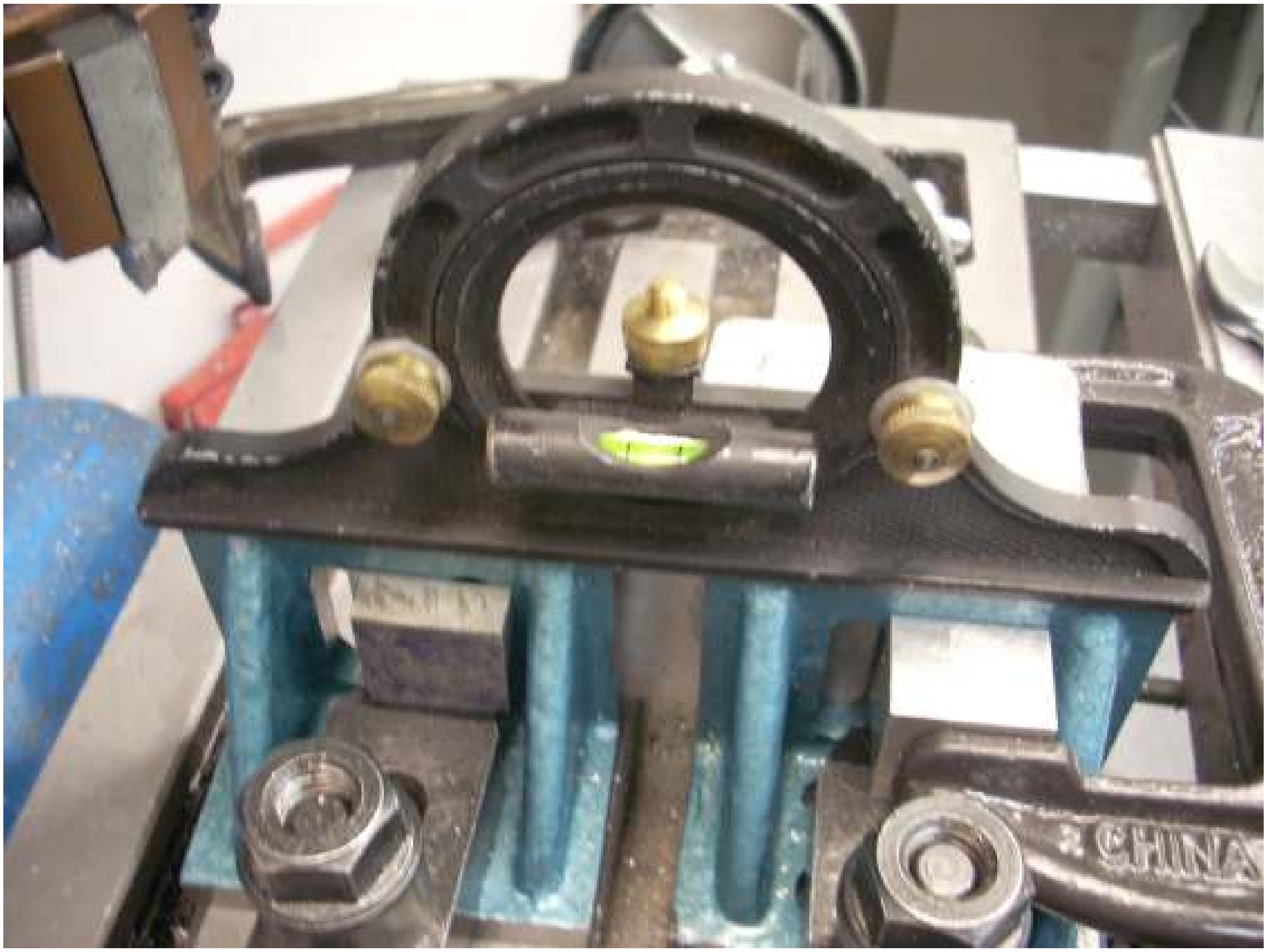
If Reference 1 secondary was critical, I would turn the casting over again with Reference 1 primary again down on the support blocks and re-cut Reference 1 secondary.

Cutting Reference 2



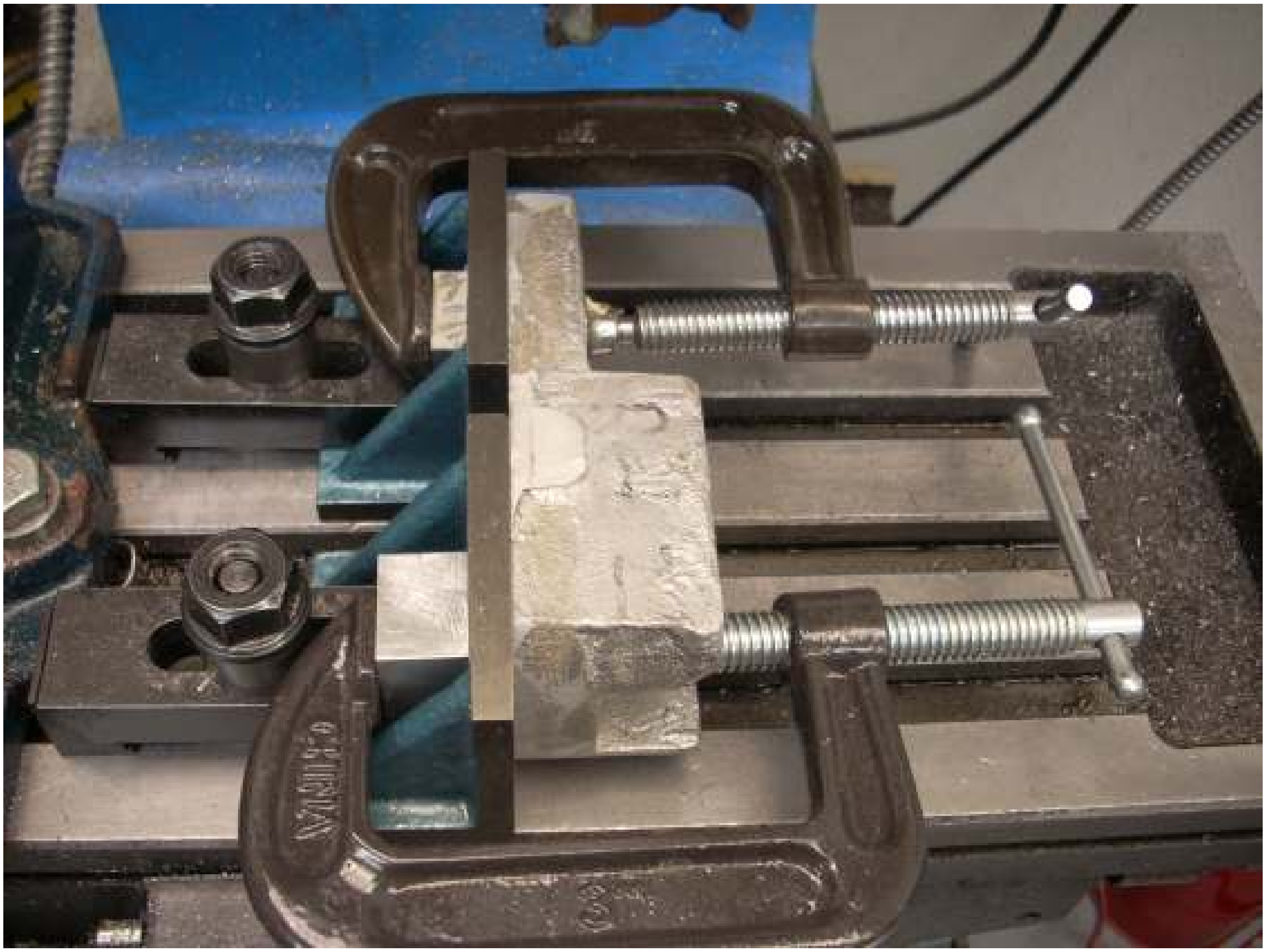
Reference 1 primary is now clamped to a knee and the second knee is secured to prevent any chance of rotation of the casting during milling. In preparation for cutting Ref 2, I must set the top face roughly parallel to the table. Since the casting is not true, much depends on setting it by eye. However, it is still good to check with a level.

Reference 2 secondary will be perpendicular to Reference 1 primary within the accuracy of the knee. If more accuracy was needed, I could clamp a machinist square to the knee and use a DTI held in the spindle to read any error. Shims could then be added under the knee to bring the vertical surface of the knee closer to true.



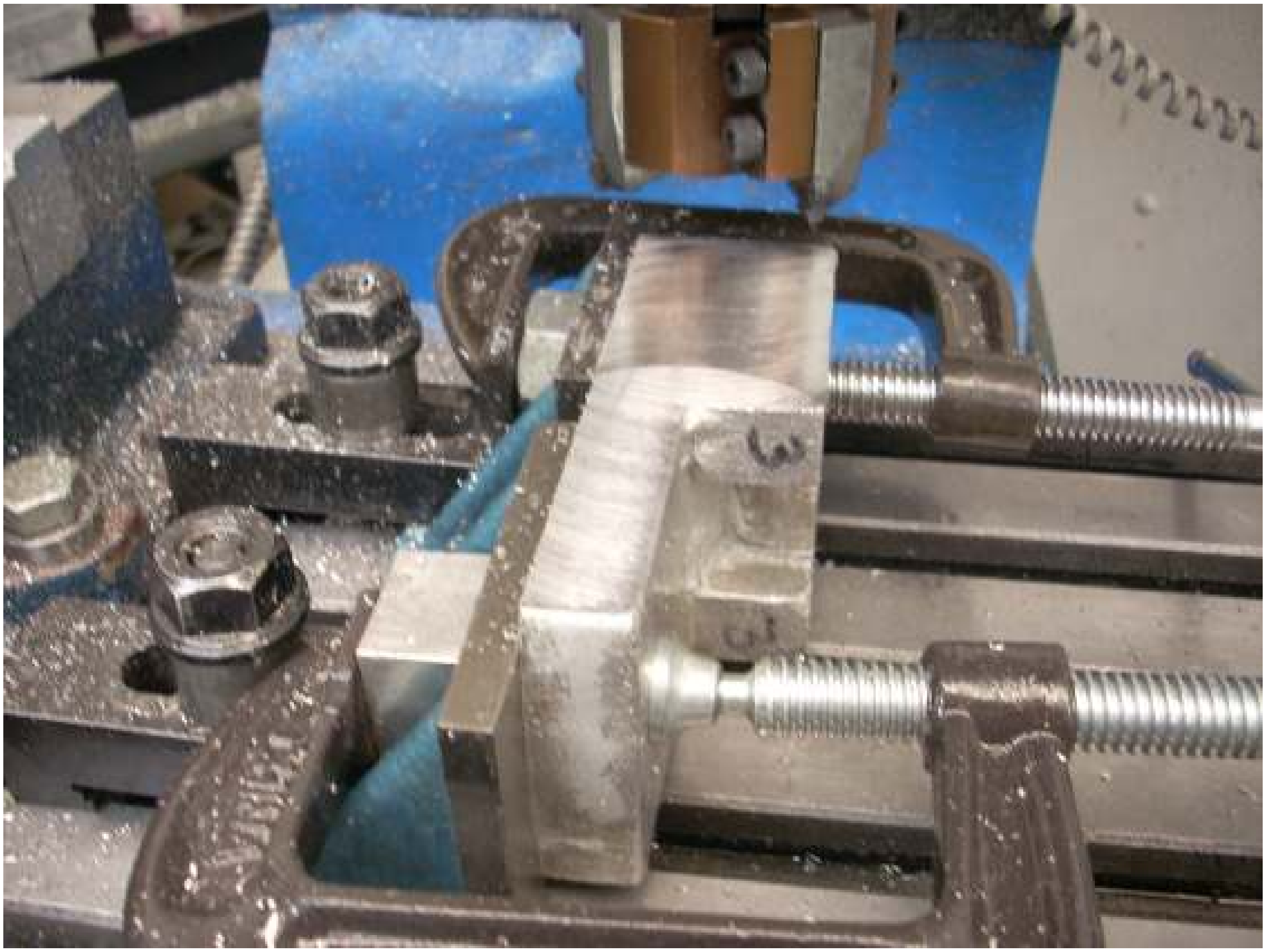
I used an adjustable level here because my mill is not perfectly level. The adjustable level was first set to show level when resting on the top of the knees.

I then moved it over and placed it on the smoothest part of the casting. My Reference 2 primary face was not touching the mill table. This permitted me to adjust the orientation of the casting in preparation for cutting Reference 2 secondary.



Once all looks close, C-clamps were tightened.


Reference 2 secondary has now been cut.


Reference 2 primary is done next. Reference 2 secondary is resting on a parallel so that the shell mill does not hit the top face of the knees. The casting is solidly supported at Reference 1 primary and Reference 2 secondary as I cut Reference 2 primary.

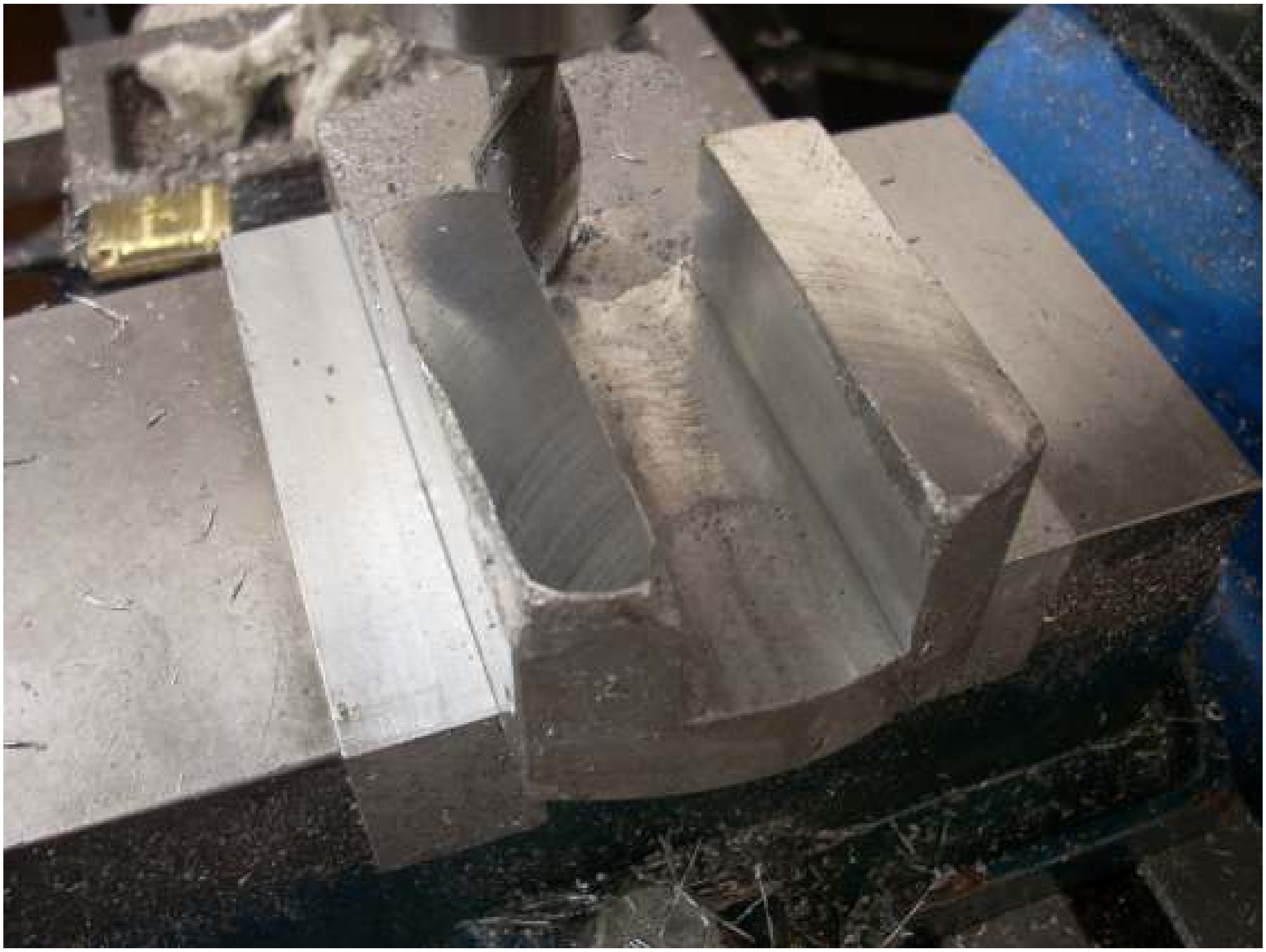
Cutting Reference 3


Reference 3 primary will be side milled with my 5/8” end mill. I'm using lots of WD-40 as my cutting fluid and taking 0.010” deep cuts to minimize bending of the end mill. My last pass is without any in-feed in order to minimize the deflection of the cutter.

I could have put the casting on end with Reference 3 primary horizontal and used my shell mill. This would have avoided the side mill deflection error problem. Why didn't I do that? Because I just realized this option as I was writing the last paragraph! Fortunately, Reference 3 primary is not used for alignment on this part.

Note that the casting is in my vise's soft jaws so Reference 3 primary should be perpendicular to Reference 1 primary and 2 if there is no end mill deflection.

Pivot Block Cleanup



Time to clean out the mess between the pivot blocks. A few passes with the end mill did the trick. I then side milled the left inside face. The thickness of this block was measured and the right inside face was cut to match the thickness. This was done just for looks. As long as the inside faces are vertical and parallel to each other, the clapper should move correctly.

As with the cutting of Reference 3 primary, I took light cuts with the last pass taken with no in-feed.

The only way I can think of to cut these inside vertical faces is with side milling. Now, *IF* I had my shaper operational, it could do a better job.

How Am I doing?

Time for a small detour. Just how true have I cut the casting so far? With the part on my surface plate I zero my DTI on one end of Reference 2 secondary. This DTI easily indicates 0.000 5”.

The DTI is carefully moved to the other end of the block. Can you see that the difference is less than 0.000 5”? Not bad if I must say so myself.

At the risk of sounding cynical, this is a classic case of “Murphy's Law”. The accuracy of Reference 1 secondary relative to Reference 1 primary does not matter at all. Had this been a critical feature, I'm sure the error would have been much larger.

Without disturbing the DTI's vertical position, the casting is turned around so the other block can be checked. This end of the block is at the same height as the other block within the sensitivity of the DTI.

The other end is 0.000 5" higher. This error probably came from an error in the support plates. For critical work, soft jaws would be used to support the casting and this error might be less. With enough effort, all "time invariant" errors can be canceled. We are then left with "time variant" errors like random vibration which cannot be canceled.

Now I'm testing how parallel the sides of the channel are with my DTI. First I set zero on the left side with the surface plate in contact with Reference 2 primary.

Running diagonally across this inside surface I get a point that is 0.001 5" lower. This is not a surprise since I was side milling. The tip of the end mill tends to bend away from the surface being cut compared to the area up higher on the end mill. I therefore get a deeper cut at the top of the end mill than at the bottom.

Back to the left end but this time near the top of the slot. I am about 0.001 5” lower than my zero set point.

My slot is about 0.001 5” wider at the top than at the bottom according to the DTI.

Now I am setting my zero with the finger of the DTI against the right block's bottom face.

At the other end of the block the DTI reads a difference of maybe 0.000 2”. Note that I did not change the amount the finger reaches into the slot so am not picking up any side milling error.

The opposing faces of the slot seem to be parallel but on cross section I have a “V”. How bad is it really? Time to bring out my spacer blocks and do a go/no go test.

These spacer blocks are accurate with ± 0.000 1”. I am looking for variations in the range of 0.001” so this is plenty good.

A stack 1.107” tall is a nice sliding fit up and down at the top end of the slot.

I can slide it up and down plus from front to back and it feels uniformly smooth.

Next I tried a stack 1.108” tall and you can see that it goes into the slot about ¼”. Say the bottom of the slot is at 1.107” and ¾” above the bottom we are at 1.108”. This is a taper of 0.001” per ¾” or 0.001 3” per inch which is not that good. If I later find that this is a problem I may have to go back and re-cut these faces using a fly cutter with Reference 2 primary and 2 secondary. I should be able to eliminate the side mill error but will pick up the error caused by changing References faces.

If the slot is too wide, the clapper block will be able to move and that will cause chatter during each cut.

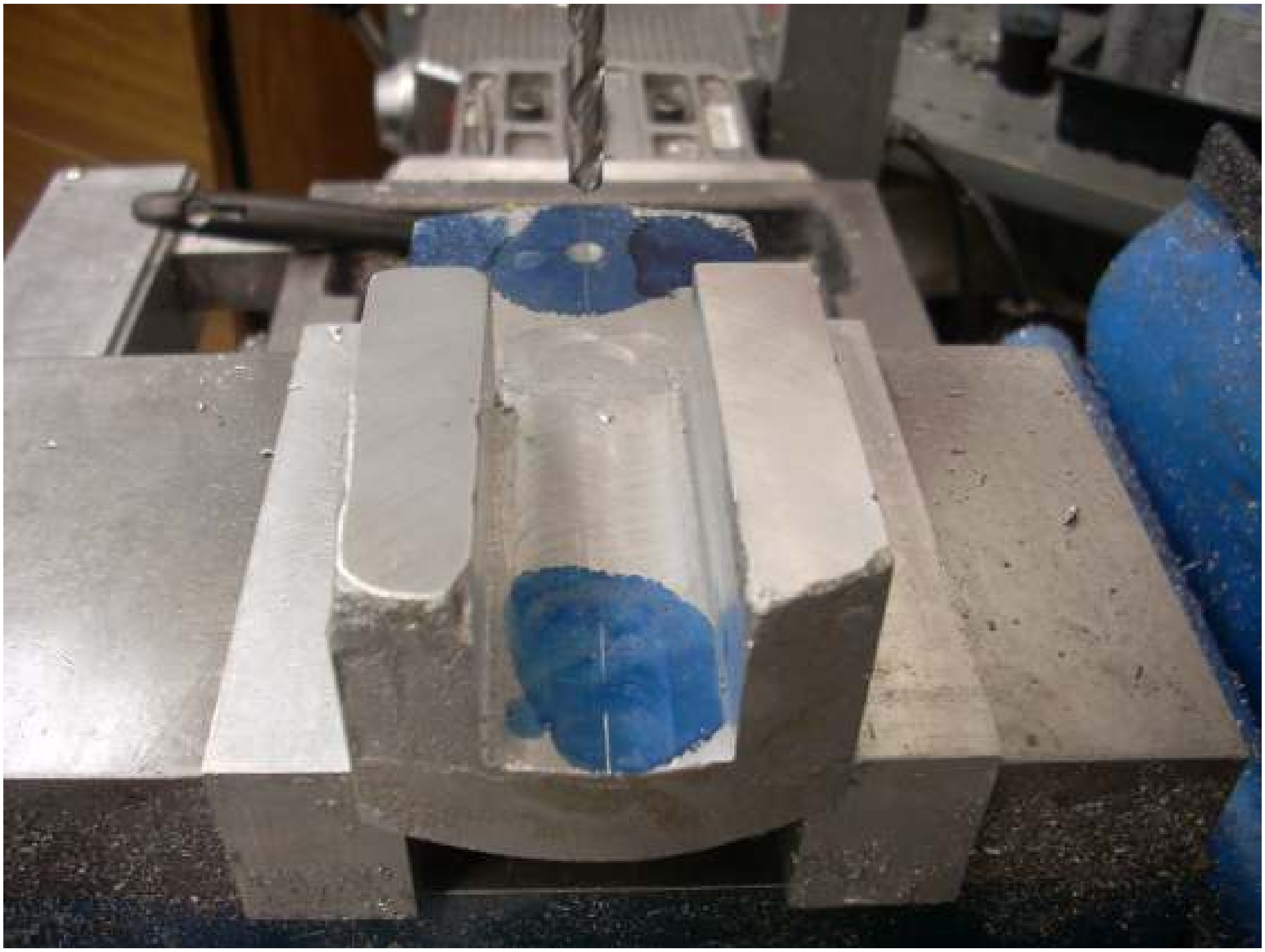
I just realized that a fly cutter might have been able to get into this slot if the casting was supported by Reference 2 primary for one face and Reference 2 secondary for the other face. Ain't hindsight great!

Another possible solution given to me by my friend Owen is to take a pass with the end mill set to side mill only the top 1/4” of the face. Then feed down 1/4” and take another pass. Continue to take passes until the entire face has been cut. In this way the end mill never sees the full force of cutting the face.

A third possible solution is to blue the out of square faces and then lightly file away the bluing while checking with the spacer blocks. This idea came from my friend Roy. I think I will try this idea first.

I went with this last approach and it is documented in my next article.

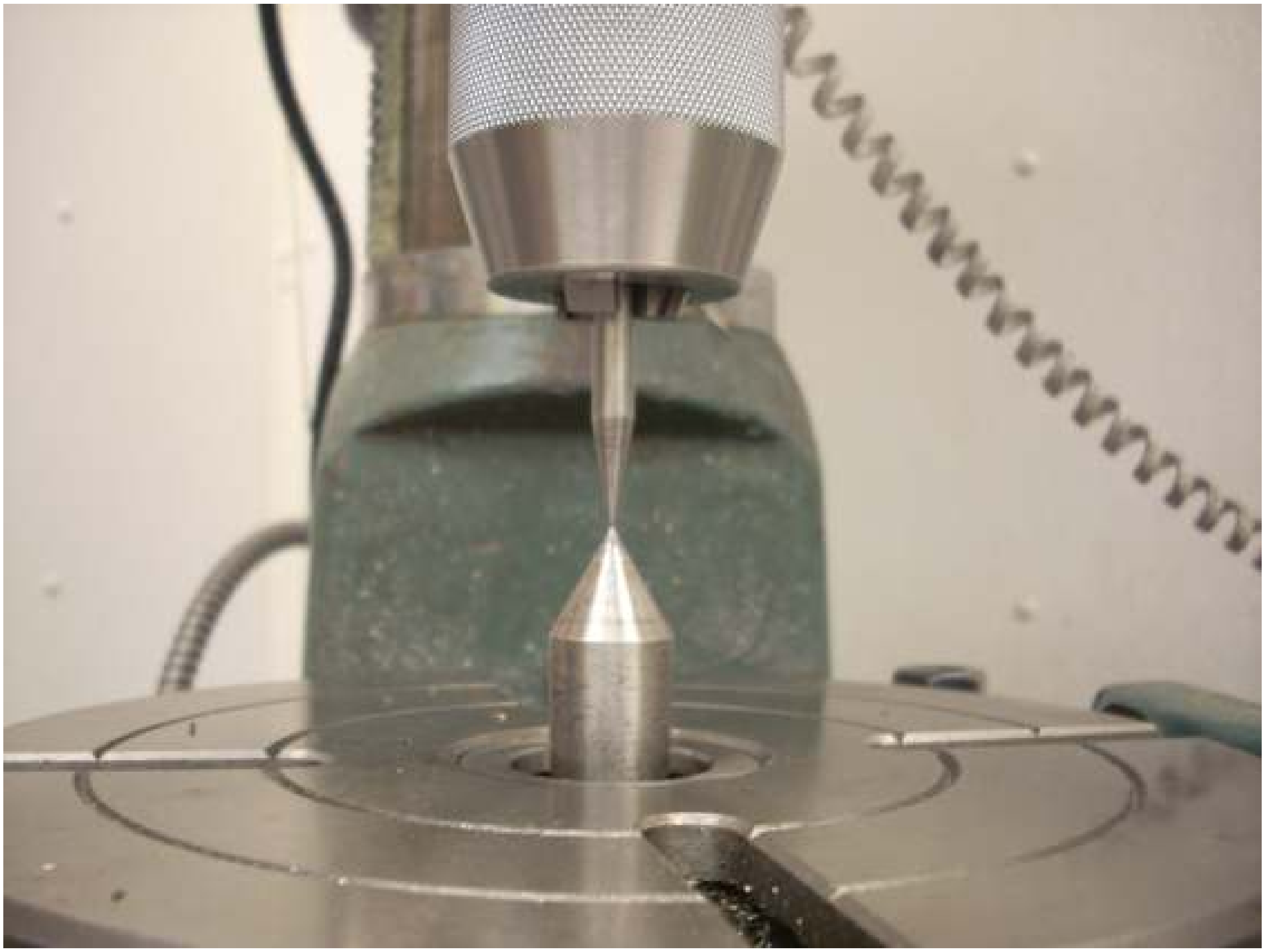
Finishing The Clapper Box



So much for metrology. Time to drill the mounting holes. In the foreground is the location of the pivot hole. I have just drilled a hole that will line up with the curved slot to be machined later.

Gingery specifies that these holes should each be ¾" from the ends but later says they should be 4" apart. Be careful with this over constraint of the hole locations. I did place the center of the holes ¾" from the edges but the center to center distance was 3.888" and not 4". This dimension is not critical but it is essential that the holes in the clapper box perfectly match the holes in the down feed casting. Not a problem if you match drill.

The holes were drilled and then the casting put aside so I could set up my rotary table. I will later match drill through these holes and into the down feed casting.



I can't recall the last time I used my rotary table. Nothing against this fine tool, I just don't find myself making things that need it.

The first step is to roughly align the center of the spindle to the center of the rotary table. I used a dead center in the rotary table and a spud in my drill chuck. This will get me close enough that I can next use my DTI.

With the Z axis locked, I zeroed the DTI at the front position[2] . The spindle is then rotated so the DTI is at the back position. The reading is noted and I adjust the Y axis until the DTI now reads half of its previous reading. I then move back to the front to verify that there is no difference between front and back reading. The process is repeated between left and right.

2 There is some risk that indicating on this steep taper causes me to pick up additional error from an out of tram head. It might have been better to put the DTI on the less steep taper or better yet, with another rod that has no taper.

When the center of the spindle is aligned with the center of the rotary table, I can can swing the DTI all the way around and the needle does not move. The X and Y DRO displays are then zeroed.

This is far more accuracy than needed for the next operation but I was a bit rusty using my rotary table so this was good exercise for my aging brain.

I adjust the X axis position until my ¼” drill enters the center hole without bending. The X and Y axis are now locked.

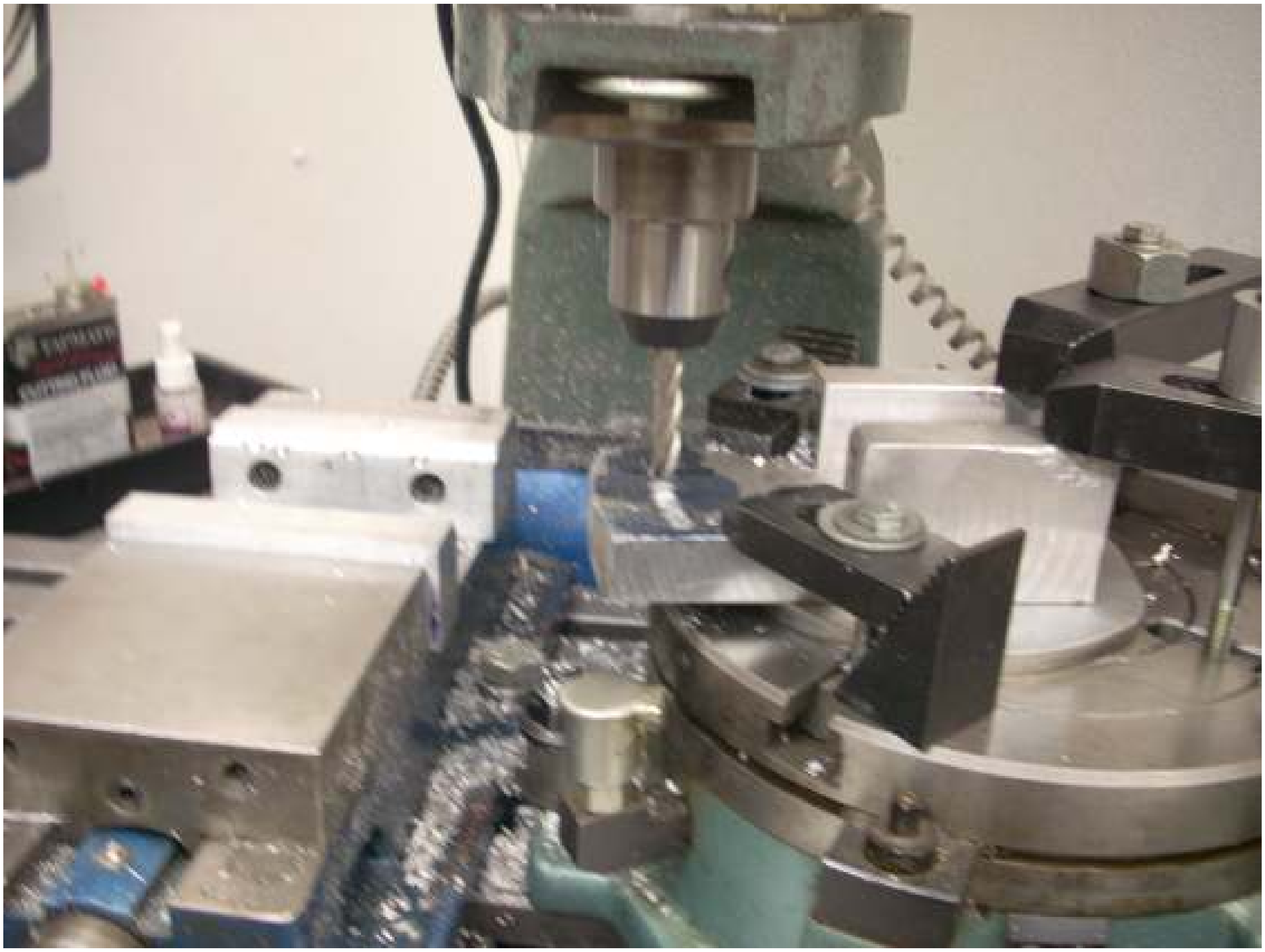
I found that a rotation of ±14° relative to the center hole looked about right.

First the slot will be chain drilled and then cleaned up with a 3/8” four flute end mill.

Here you see the first hole being drilled at about 13°. That leaves some metal for my clean up cut with the end mill.

I chain drilled from that first hole towards the center hole.. Then I started a second series of holes starting at +13° from the other end of the arc.

The drill and drill chuck have been replaced by the 3/8" four flute end mill in a collet. This type of end mill cannot drill like a two flute. Instead I have centered it over a hole which provides relief for the center of the end mill which cannot cut. Note that the hole is only part way through. Yup, screwed up again – I ran out of quill travel.



I raised the quill all the way up and then unlocked the mill head. After lowering the head down, the quill was extended until the end mill was near the surface. The head was then slowly rotated until the end mill lined up with the hole. As a final check, the quill was lowered to verify that the end mill cleanly entered the hole. Then the head bolts were tightened.

Here is the root of the problem. The 1/4” drill plus chuck are much longer than the 3/8” end mill and collet. Fortunately, this is not a critical cut although my alignment is probably within a few thou. For better alignment I could have run my DTI around the inside of the hole milled by the end mill and achieved better alignment

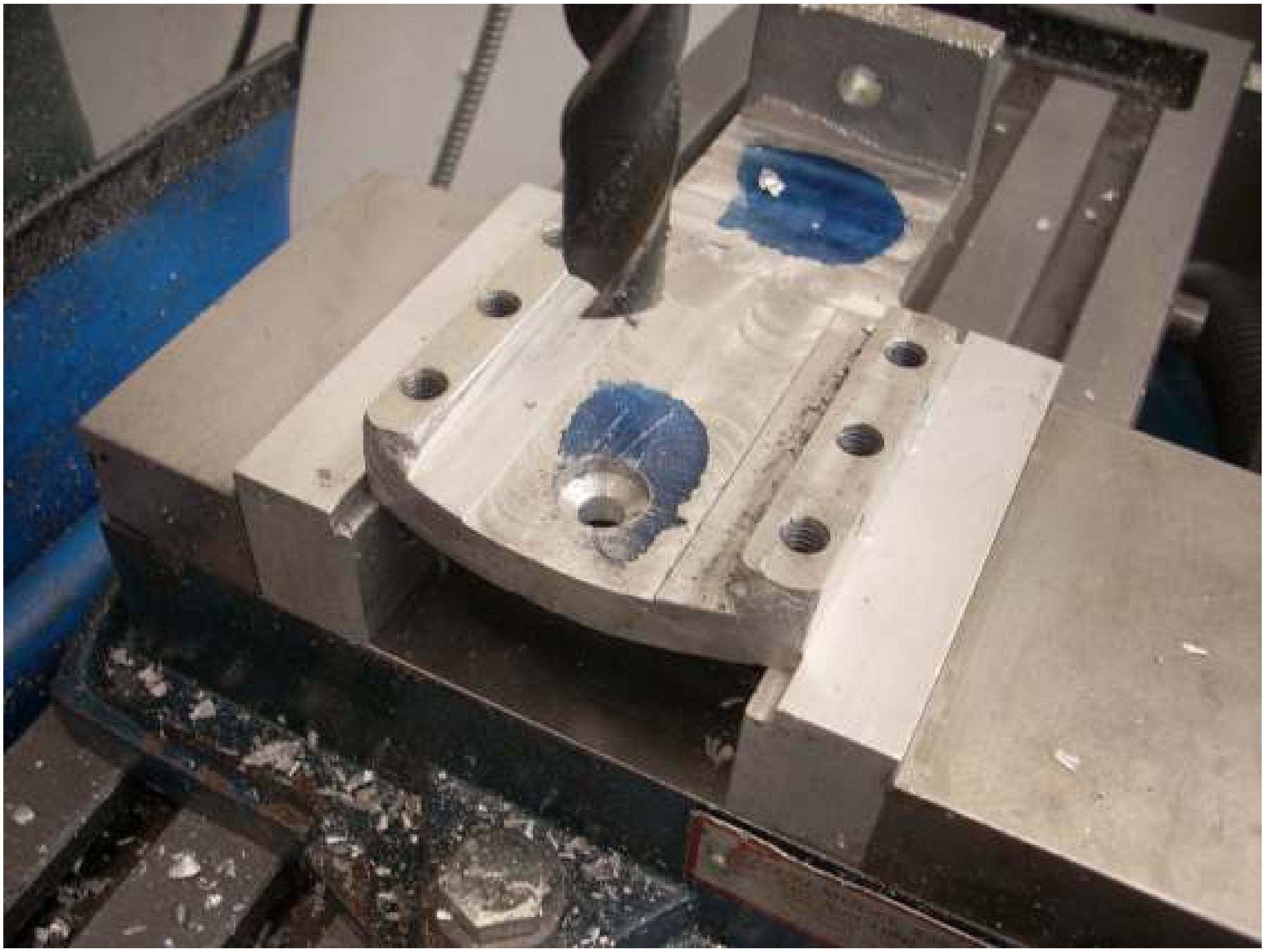

The slot is then side milled. I moved the rotary table from -14° to +14° and get a nice looking arc.

Looks good to me but the top surface is a bit ratty. Gingery notes that I can optionally clean up the surface. I offset the X axis + and – 0.200" to clear an area around the slot.

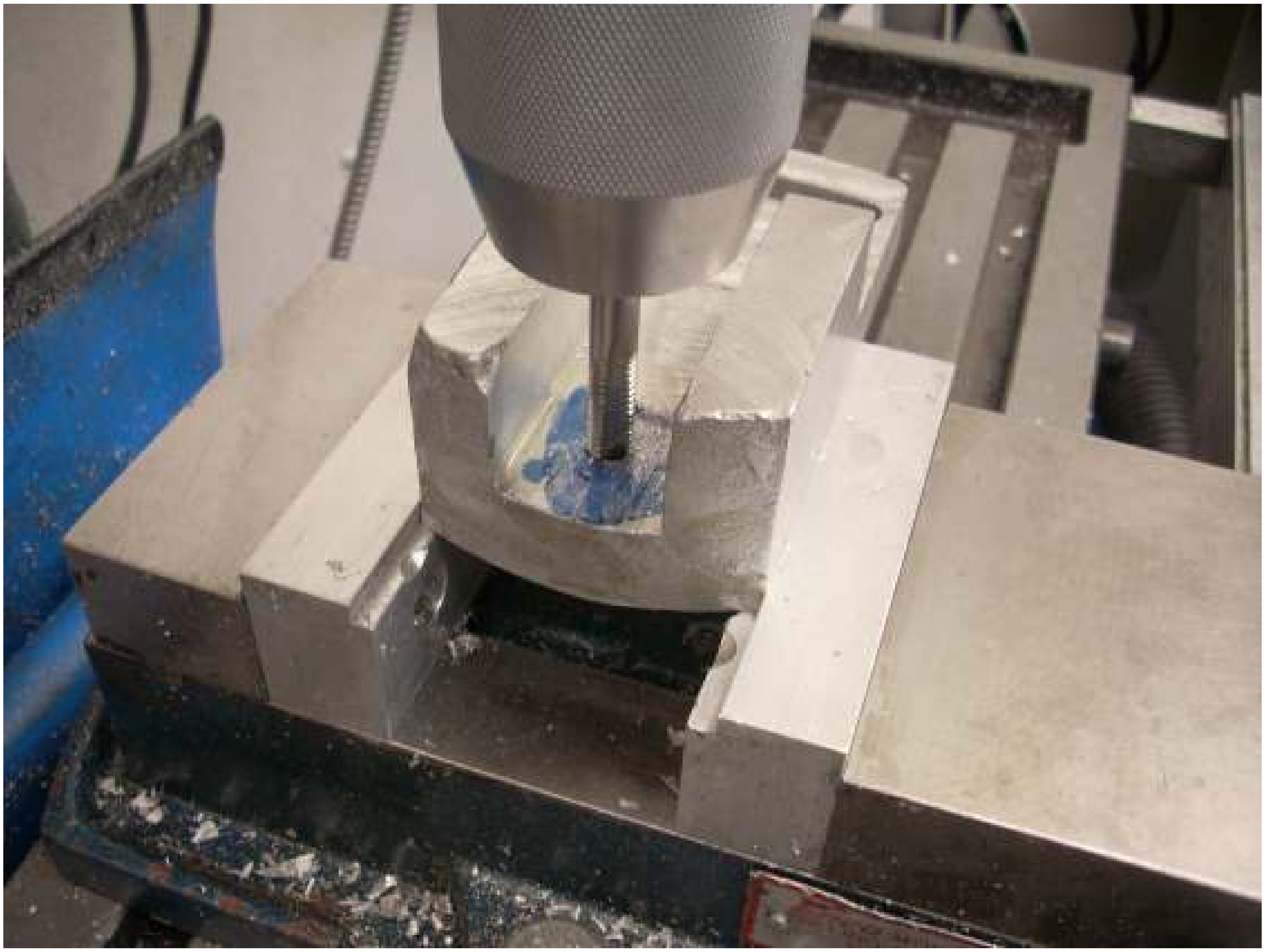
I'm not real happy with the look, especially at the ends of the arc.


The part was put back in my vise and the surface milled.

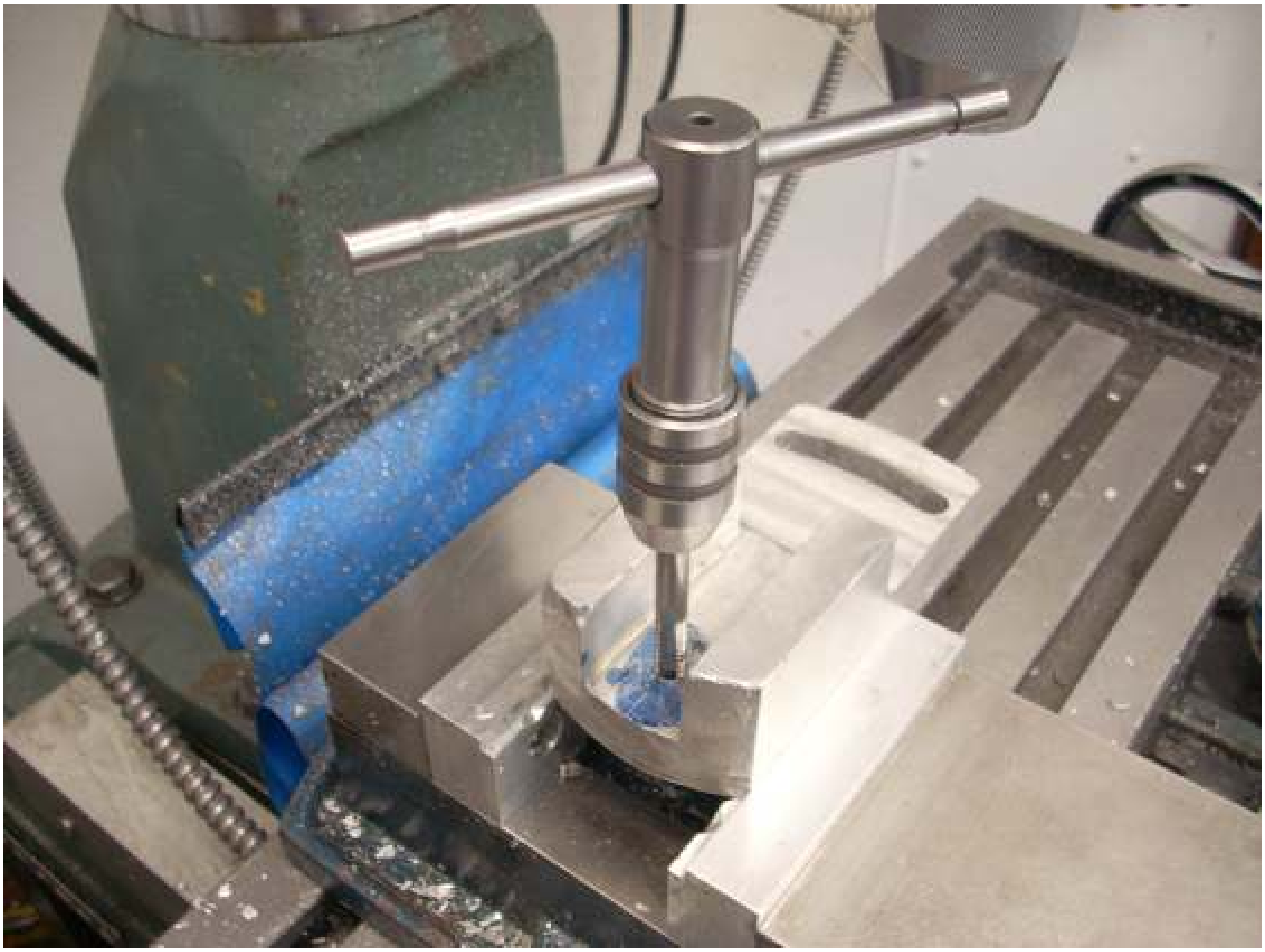
Final Machining of the Down Feed and Clapper Box



Back on the down feed casting, it is time to machine the pivot hole. I have drilled a clearance hole and sort of a counterbore using a 5/8” drill. I wanted to use my 5/8” end mill but it would not reach without lowering the head. This will work fine.


The pivot hole must be tapped in the clapper box.

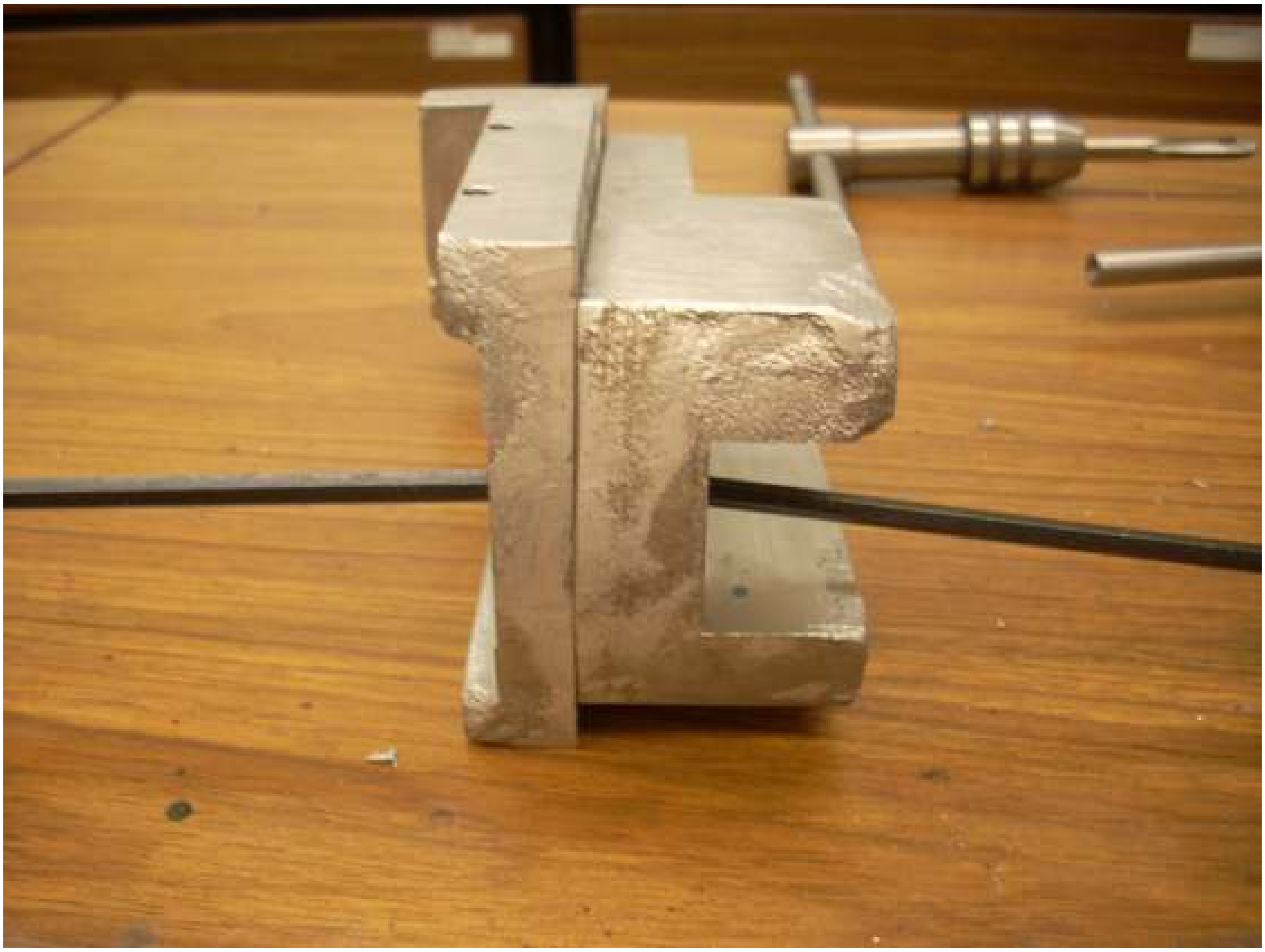
With a spiral point tap in my drill chuck, I ran the mill up to full speed and then cut power. After it coasted down to a comfortable speed I drove the tap into the hole. It went in about half way which is safe.



The thread was finished up by hand. I am too far along to risk breaking off a tap, stripping out the threads, or yanking the casting out the the vise.

If I had striped out the threads, a Helicoil® would have been used to restore the thread.

The clapper box has been temporarily bolted to the down feed casting and set approximately on center. I can now match drill the clamp bolt hole such that it is aligned by the curved slot. A spud was used to position the center of the spindle on the center line of the down feed casting. Next a close fitting dowel was used to locate where along the X-axis to drill the second hole such that it is centered in the arc. The hole was then drilled and tapped.



A screw is fed through the down feed casting and into the threaded hole in the clapper box. A set screw is fed in from the clapper box. By tightening the screw from the left and the set screw from the right, their ends press together and lock the screw. The idea is to have an adjustable clamping action between these two parts yet permit them to pivot.


With the screw installed, you can see that I am able to swing the clapper box all the way to the left.

I can then smoothly swing the clapper box all the way to the right.

The bolt head shown above was carefully chosen to be the same size as all adjust bolts on the shaper. I don't want to have to constantly swap out sockets while setting up the machine.

One last bit of clean up so the dial feed pointer and crank fit squarely on the above surface. I could have cut a bit deeper and not left the small recess but did not want to push my luck with the casting after so many hours of time invested.

We are now one step closer to having an operational shaper!

The clapper will be cut from CRS. I plan to use a tapered D reamer to cut the pivot pin hole. The tapered pin will be cut at the same time the reamer blank is cut. But that is for another day.

Rick Sparber

rgsparber@AOL.com