NAPBIRT TechniCom

Sharing the Craft for Over 35 Years

Western Iowa Tech Community College
Band Instrument Repair Class 2013-14

The bimonthly publication of the National Association of Professional Band Instrument Repair Technicians, Inc. (NAPBIRT)®

Volume 38, Number 6
November-December, 2013

Saxophone Post Fabrication Using Form Cutting Tools *By Mark Nagy*

Recently, I acquired two saxophones that I wanted to use as loaners. One was a Yamaha 21 alto and the other, a Vito tenor. Both were missing some posts: the alto, a side F# post and the tenor, a G post and high E post. Due to their age, many of the parts are no longer available for these saxophones. My first thought was to modify an existing post to fit the saxophones. Unfortunately, none of the posts I had, for one reason or another, would work. After thinking about the situation, I decided to fabricate some posts using a form cutting tool. I will describe the process for making the post for the alto. I did the same for the tenor, but with different dimensions.

First, I needed to create a tool holder. Since I needed to cut several posts of different sizes, I needed a tool holder that would allow different cutters to be used. I made the tool holder out of 01 steel. The surface that would be holding the cutter in place was machined down on a vertical mill, to make sure that the edge of the cutter would be in contact with the center of the work piece. One drawback is that this holder is made for the height of the tool post of my lathe and if I try to use it on another lathe, it may be at the wrong height. To hold the cutter in place, I used two 10-32 ½" socket cap head screws.

After making the tool holder, the next step was to draw a sketch to help me determine the dimensions of the new posts. I measured the post that was still on the body of the saxophone to create its match. After I made the measurements and created a sketch, I was ready to make my cutter.

I used .025" x 1.00" 01 steel (available from McMaster-Carr). In order to make a tool that would cut effectively, it would need to have the proper relief, which means that the cutting surface must be the first and only part of the cutter that would contact the work piece. Otherwise, a part of the cutter, other than the cutting edge, would contact and rub against the work piece, which means no material would be removed. A cutting tool needs relief of at least 10%. I used two 15% angle blocks to hold the steel in a machine vise, while I used a center drill and then a drill bit to cut the holes which would form the contour of the ball. I cut the holes near the edge of the steel, with the idea of removing the half of the hole wall on a bench grinder, leaving half a hole for the diameter of the ball. Initially, I tried a 7/32" bit, but the contour was a little too big, which would create a larger ball than I desired. I settled on using a 3/16" bit (.1875")

After I drilled the holes, I removed the excess steel to form the contour of a ball. I tried to cut the relief by holding the cutter in my hand, but I found that when I needed to cool the work piece, in water, as it got hot, it was difficult to find the same orientation on the grinding wheel. To solve this problem, I mounted a grinder tool rest (item 05M23.01) available from Veritas (www.veritastools.com). This way I could always return to the same position, in terms of the relief angle.

After getting the rough shape of the cutter, I needed to finish shaping and making a relief for the hole. For this, I used a stone grinding wheel in a Dremel tool. I found that a wheel in the shape of a ball seemed to work the best. The item number is B122 available from Victor Machinery (www.victornet.com). This company has a variety of tools and accessories for machinists. The diameter of the ball was 3/8", which is slightly larger than the size of the ball contour on the cutter. I ground material from the bottom of the tool cutter to the top, stopping just before I reached the edge of the cutting surface. This made sure that everything below the cutting edge was lower in terms of the tool profile, making sure that I had the proper relief.

Next, I made an angle, to shape the angle of the post. Again, I used a bench grinder to cut this edge. Before heat treating the cutter, I tested it by cutting an old drumstick. Even though wood does not have the tensile strength of metal, it was strong enough to let me test the shape of the cutter. The drum stick eventually broke off, but I cut enough material to get a good idea what the shape of my post would look like. I realized that I needed to modify the angle of the post, as well as the size of the base. One thing to remember is that, on a form cutting tool, everything is in reverse of the shape of the final product; the more something protrudes on the cutter, the more material it removes. In my case, I found that the base of the post was too wide, meaning, it was removing too much material from the area near the base of the post. To change this, I needed to remove more material from the area of the cutter near the ball. (see Diagram) As a result, the ball became smaller and I needed to enlarge the hole. After modifying the cutter, I tried it again on another piece of wood. Now, the cutter produced a post in the shape that I wanted.

Next, I needed to hone the cutter. Since the through holes created a burr on the bottom of the cutter when they were drilled, the cutter would not sit flat in the tool holder. To remove this burr, I used 120 sandpaper placed on a flat surface, like a drill press table, to flatten the top and bottom of the cutter. Next, I honed the cutting surface with an India sharpening stone made by Norton (model IB6). This stone has a coarse and a fine surface. After using both surfaces, I then used an Arkansas stone, made by Hall's ProEdge sharpeners (www.hallsproedge.com). This stone also has two surfaces, hard and soft. Again, I used both surfaces to finish sharpening the cutter.

After honing the cutter, it was ready to be heat-treated. I used MAPP gas to heat the cutter until it was cherry red and dunked it in water. Now, I was ready to try to cut a post. I decided to do a test cut on a piece of 6061 aluminum because it is cheaper than 360 brass. I set my lathe to its lowest RPM and began to cut. Most cutting operations on a lathe remove only a small bit of material (usually only a few thousands of an inch). However, I was trying to remove about ¾" of an inch of material at a time. So I needed to remember to go slow. I set up a live center to support the work, from the right side of the cutting tool, to minimize flexing. I discovered that my cutter was cutting; however, it began to dull in one area. I realized that I didn't heat the tool evenly when I treated it. I remembered that as I moved the flame from one area to another area it would start to cool. Also, I held the cutter with a pair of pliers, which acted like a heat sink and made it difficult to heat the cutter cherry red. I re-honed and reheated the cutter, this time using two torches; one torch was on the bench and the other was in my right hand. I suspended the cutter from a piece of wire, over the water, with my left hand. This time I heated the surface, consistently and it seemed to cut the aluminum better. My cutter looked like it was going to work! (see Photo A) After my trial on the aluminum, I cut a post out of ½" 360 brass. Before I started to cut, I placed the cutter as close to the jaws of the chuck as possible and locked down the apron screw to avoid any lateral movement as I cut the post. (see Photo B) Even with the support of the live center, the work piece was flexing and as a result it made a lot of chatter. I went very slowly, moving the in-feed only about .002" or .003". I watched, listened and felt how the cutter was working. If I felt it was going smoothly, I kept the same rate of in-feed. If I felt that the machine was laboring, I slowed down the rate of in-feed to allow the cutter to catch up. I needed to use a lot of cutting fluid and periodically turn the lathe off to allow the part to cool for a while. You have to remember that you are trying to remove a lot of material at once. I decided to cut several posts as long as I had the lathe set up to do this operation. I figured I could use them, sometime in the future.

After I successfully cut several posts, I needed to cut and thread the holes for a pivot screw. To do this operation, I made a holding jig out of 1" square 01 steel. (see Photo C) The idea of this jig is to have the remaining ½" brass stock on either side support the post as I cut the holes and minimize flexing, like a live center supports a work piece in the lathe. However, as I discovered when cutting the hole, there was still some flexing. First, I cut a .500" through hole in the center of the stock, used a .500" reamer to clean the hole, and followed with a .501" oversized reamer (Micromark item 84753) to allow the jig to receive the .500" piece of brass. I then drilled and tapped a set screw to keep the work stationary while I cut the holes. I used a 5/16-18 set screw. Next, I cut away a portion of the jig, to expose the work piece, in order to cut the pivot screw hole.

To cut the hole in the post, I first used a small file to make a flat spot, on the ball surface, because it is easier to cut a hole on a flat surface than a curved surface. I used a number 1 center drill, to make a small divot for the drill to enter the brass. Next, I needed to modify the drill bit I was going to use to cut brass. I flattened the cutting lips, on the side of a grinding wheel, to allow the drill to scrape the brass rather than shear it. If you use a regular jobber drill bit on brass, it will grab the material due to the rhombus shape of the brass lattice, compared to the cubic shape of the steel lattice. I wanted to use Yamaha pivot screws, which are 3.0 x .5mm. To determine which drill to use for a metric thread, you subtract from the major diameter the distance between thread crests. In this case, it was a 2.5 mm drill. I didn't have a 2.5mm drill bit, so I used an Imperial equivalent. Since 2.5 mm equals .0984", I used a number 40 drill bit, which measures .0980". I found it difficult to put the tip of the drill bit in the divot, so I used a series of smaller drill bits, to work my way up to the number 40 bit. I drilled and tapped the hole using the jig. I then put the post back into the jaws of the lathe and cut off the excess brass that was used to support the post in the live center and used a file to smooth over the surface of the ball while the lathe was spinning. Originally, I wanted to have the cutter completely shape the ball, but I realized that I needed the support of the live center to avoid deflection. At first, I wasn't too happy with the idea of having a piece of brass, that would be wasted, but I realized that, even though it was a small amount, I could use it for a later project.

I was now ready to fit the post, to the body of the saxophone. I started by using a cylindrical sanding wheel in a Dremel tool. Once I had the rough shape, I held the post on the body of the saxophone, as I pulled a piece of 120 grit sandpaper under the base of the post, to match the radius of the saxophone body. Finally, I soldered the post on the body and the saxophone was now playable.

I repeated the same process for the other posts. Since the high E post had two balls on it, it was more challenging. It was very difficult to make both of the balls match in terms of size. When I cut the post, I realized that one was .012" bigger than the other. Instead of making another cutter, I simply used a tonehole file, to remove the excess material, as the work was spinning in the lathe.

The process of making these cutters was long. I spent my weekends and free time during the slow winter months working on them. At first, it took several hours just to make a cutter. Now I can make one in less time. During the process of making the posts, even though I made sketches and had a plan, I needed to be flexible. I ran into unforeseen surprises and needed to come up with solutions, for example, when the first cutter wasn't treated thoroughly. Also, there was a certain element of trial and error, especially when I was making the contours of the cutters. I had never made a cutter before, so I needed to have an idea, try it out and see what the result was. I learned quite a bit during this process, especially with regard to the reliefs needed to make a cutter that will work well. All one really needs to make a project like this one a success is patience, perseverance and a lot of 360 brass stock.

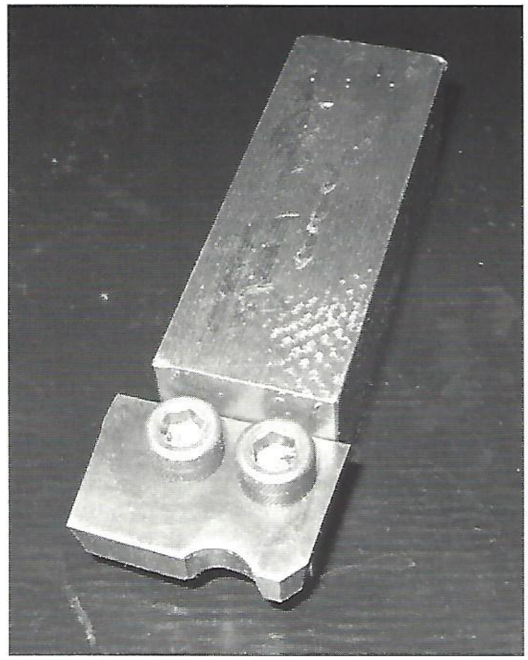

Photo-A

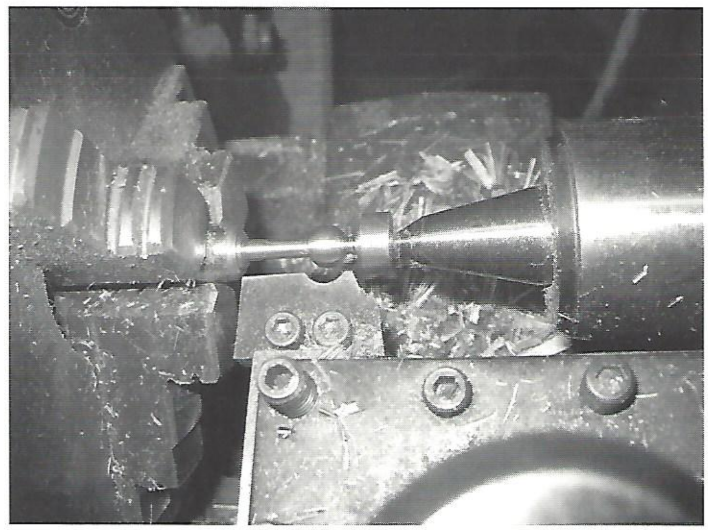

Photo-B

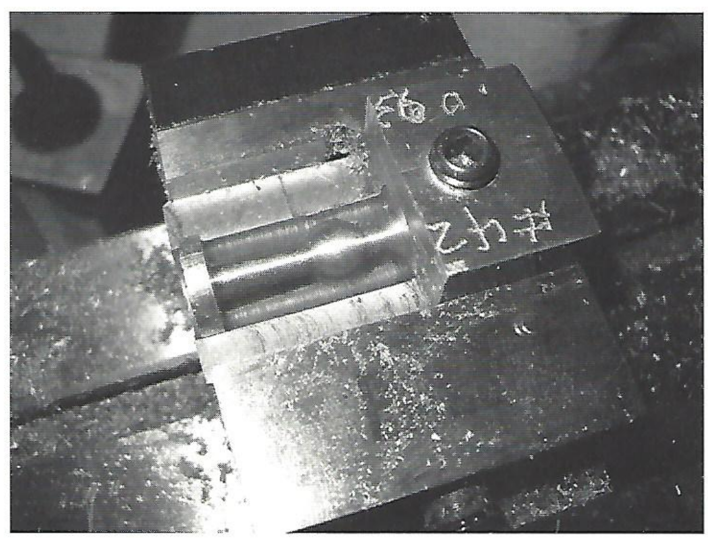

Photo-C