Making Threaded Rivets, Version 1.3

By R. G. Sparber

Warning: From a safety standpoint, limit the bolt size to no larger than #10.

Once in a while I need to have a threaded hole but the material is too thin for tapping. That is where a threaded rivet can save the day.

Here you see 0.054 inch thick aluminum stock. The screw is 6-32. At 32 threads per inch, I get one thread per 0.031 inch. I need at least 3 threads to hold the screw so tapping a hole won't work. I would only get $\frac{0.054}{0.031} = 1.7$ threads. Yet this screw is secure because the threaded rivet has a threaded section about 3/8 inch long with over 12 threads.

Commercially, these threaded rivets are called Rivnuts and many videos on YouTube can show you how to ***install*** them. Didn't see any that told me how to ***make*** my own. Furthermore, they have a limited crimp range. I found some for 0.125 to 0.128 inch thick material which means a washer would have to be added here. Given my 0.054 inch angle stock, I would need to find, or more likely make, a washer 0.074 inches thick with a specific inside diameter.

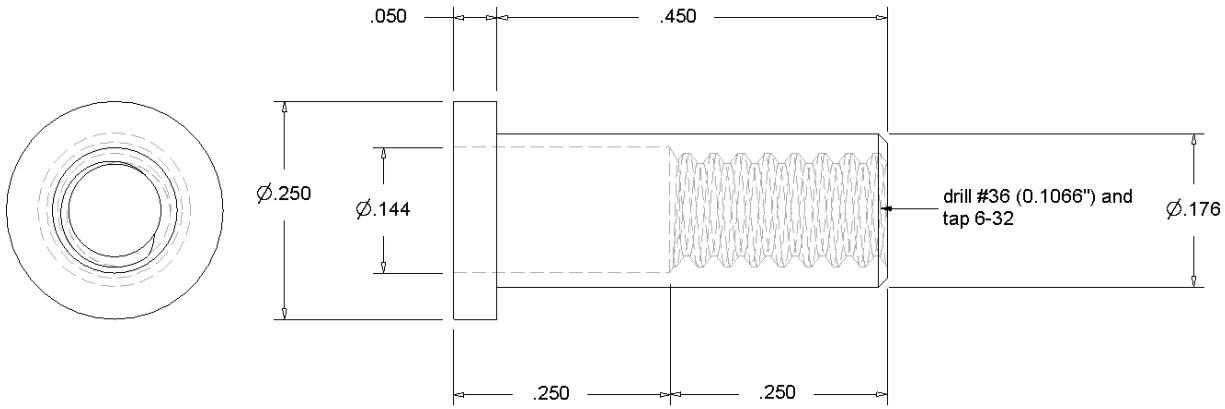
Well, it turns out that I can make these rivets for not a lot more effort and pick my crimp range so no washer is needed.

This is a good project for someone new to machining. You will need some aluminum rod and a lathe plus some common cutters, drills, and taps. If you don't have any of these, this project will give you the perfect excuse to go on a buying spree.

A nice animation of how a threaded rivet works can be found at

https://www.youtube.com/watch?v=dkPhGIaPCVs

I will walk you through the making of a 6-32 thread designed to pass through 0.054 inch thick stock. All of the work is performed on a lathe. Later the logic needed to make any size threaded nut will be discussed.



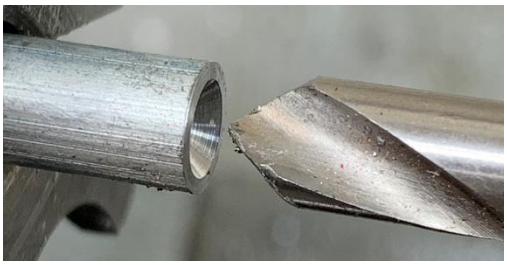
It is essential that the rivet be a snug sliding fit in the receiving hole. Here I have drilled a #16 hole in a piece of scrap 1/8 inch thick aluminum plate. This gave me a nominal 0.177 inch diameter hole. It is my gage to size the rivet's diameter. As long as I use the same drill on my project, all will fit nicely.

On to making the threaded rivet!

I chose to use extruded aluminum rod for this rivet because it does not have to be very strong to support a 6-32 screw. The metal is soft which will make the crimping operation easier. Tempered aluminum is more likely to crack.

"Toolmaker51" on homemadetools.net pointed out the following:

"Generally, certain Aluminum alloys can be annealed easily. 'Brush' the part with a neutral oxy-acetylene or propane torch and air cool. This is a frequent occurrence in metal spinning."



I chucked up a length of extruded ¼ inch diameter aluminum rod. After facing the end, I drilled it with a ¼ inch spot drill. This will ensure that my drill centers in the rod and does not wander.

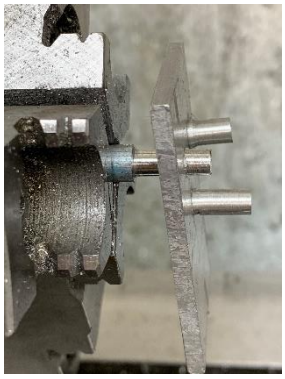
The stock stuck out about 0.6 inches.

I prepared to cut a stop mark ½ inch from the end.

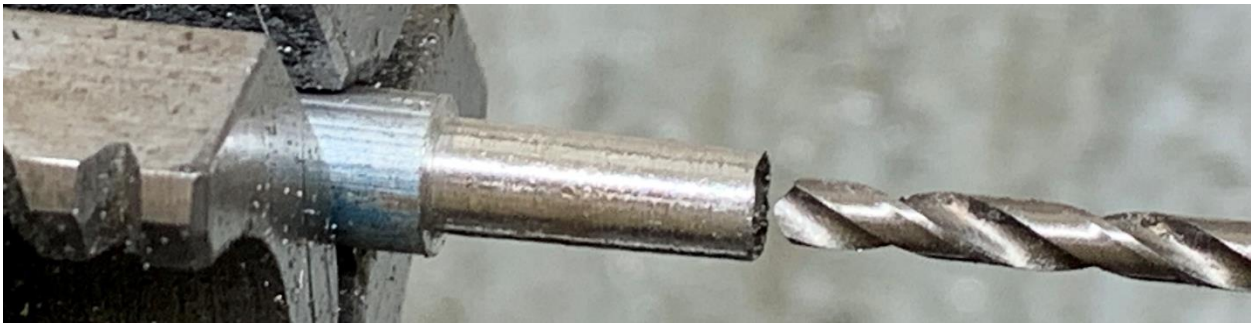
I also touched the cutter down on the rod's surface and zeroed the crossfeed dial.

Next, I zeroed my calipers with the jaws closed. Then I opened them to 0.176 and zeroed the caliper again. This told me how far I was from my target diameter.

With the caliper on the rod, I could see that my diameter was 0.0655 inches larger than my target. This meant I must feed in about 0.033. My first pass was 0.015. I measured again and adjusted my second pass to be slightly larger than the target.



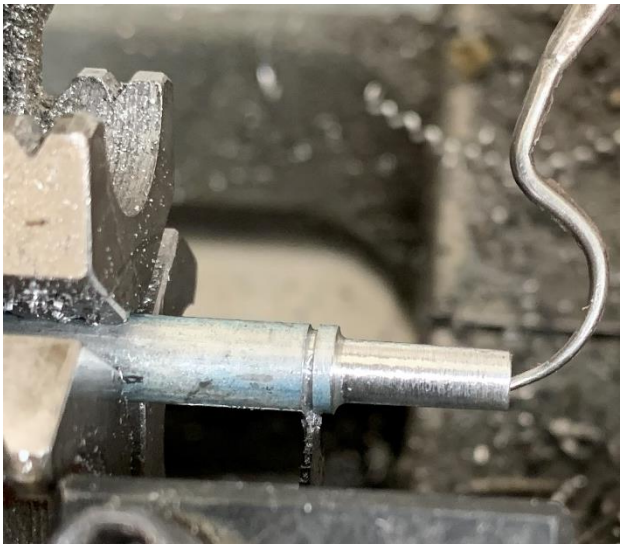
Using my gage, I took a series of light cuts so I end up with a snug sliding fit.



I then used a #36 drill which is the tap drill for a 6-32 thread. Fed in the drill about ¼ inch using plenty of cutting fluid. Then retracted the drill and cleaned out the flutes. Went in a second time for a total depth of a little over ½ inch. The added depth gives plenty of room for the chips generated by tapping.

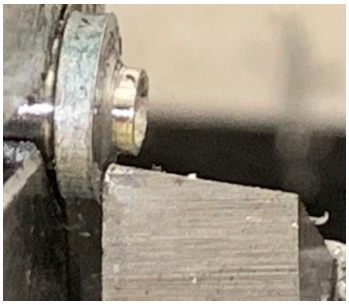
I then tapped the hole 6-32 to a depth of about 3/8 inch. Used plenty of cutting fluid.

The bottom end of the rivet was done so it was time to part-off. I used my 0.05 inch thick grooving tool[2] but went all the way through.



You can see the cut is about halfway through to the tapped hole. The dental pick gently supported the part so it would not go flying.

[2] See https://rick.sparber.org/MakingA_GroovingTool.pdf



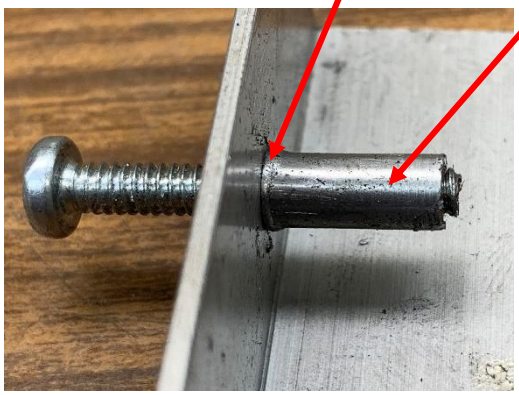
The part, back in the 3 jaw chuck, was ready to squared up the top of the rivet.

The final, and most critical step was to cut a clearance hole using a #26 drill. This is one number drill size greater than normal for a 6-32 thread. I did this to reduce the wall thickness of the section to be crimped.

Went in ¼ inch. Do not count the end of the drill in this measurement (see the shop drawing).

The wall is thicker where the threads are cut.

This means that the rivet buckled up against the supporting material while leaving the threaded section intact.



I am going to blame the poor surface finish you see here on the softness of the aluminum rod. Using aluminum with a higher temper will have a better finish but will also make it harder to form the crimp without cracking.

Installing the threaded rivet required a means of keeping the flange down on the supporting material while pulling up on the threaded section. I assembled this tool from scrap. Plenty of room for improvement.

The block was drilled with the #26 drill to clear the 6-32 screw. A nut is turned by the box wrench which presses on a washer in contact with the top of the block. Initially, the block must be held down so the threaded rivet doesn't rise up or turn.

You will feel a small resistance while the rivet buckles. Then it will be hard to turn the nut further. Back out the screw. You are done.

Scaling up from a 6-32 screw is not straight forward. First of all, I only had tempered aluminum rod above ¼ inch. This is more likely to crack during crimping. Another problem is that drill bit diameters are not evenly distributed. As we approach ½ inch, the choices are few. "Engineering Judgement" will be

needed. If you plan to make a different size threaded nut and/or use tempered aluminum, expect to make a few samples first. As long as you learn from each sample, the time is well spent.

"Kent" from homemadetools.net pointed out that there is a risk of the rivet pulling out when the stock is thin. Adding a close fitting washer to the backside will spread the force. He also recommends not using these homemade threaded rivets above a #10 screw when there is significant load.

The Design Procedure

1. Pick a thread
2. Record the tap and clearance hole
3. The Outside Diameter (OD) of the rivet equals the clearance hole diameter + 0.04 inches so the wall will be 0.02 inches thick
4. The hole in the supporting material must be a close sliding fit to OD of rivet. If necessary adjust OD of rivet to match the best available drill size[3].
5. The threaded section must contain at least 3 times threads.
6. The clearance hole depth equals 1/8 inch + flange thickness + material thickness
7. Flange OD is at least OD of rivet + 0.1 inches

Example: Just to see if I could do it, I wanted to anchor a 3/8-16 bolt in a plate 0.073 inches thick. Here you see my first try. It appears that my 0.05 inch thick flange is too thin and some of the threaded section has come through. However, the threaded rivet is solid. Obviously, the rivet cannot survive the kinds of forces a 3/8-16 bolt can experience. If I wanted to perfect this size threaded rivet, 2 or 3 more iterations would be needed.

The Actual Design

1. I selected 3/8-16 for my threaded section.
2. This means a **5/16** inch diameter tap drill will be needed and the clearance hole is cut with a **W** drill (0.386 inches)
3. The OD of the rivet is ideally 0.386 + 0.040 = 0.426 inches

[3] I bought a cheap set of metric drills. Then tend to fill in gaps in my selection of fractional, letter, and number drills. I use this combined table to find the closest size fast: https://rick.sparber.org/FastAccessDrillSizeTable.pdf

4. The hole in the supporting material is ideally 0.426 but the closest is **29/64** (0.4531). This would make the wall in the crimp section (0.4531 - 0.386)/2 = 0.0336 inches. Since the bolt is so big, it should be able to distort this thicker wall.
5. 16 TPI is one thread in .0625 inches so threaded section must be at least 0.1875 inches. Use **0.375** inches.
6. Drill clearance hole 1/8 inch + 0.05 inches + 0.073 inches = 0.248 so go with ¼ inch.
7. Flange OD is at least 0.422 + 0.1 = 0.522 inches. If I used ½ inch stock, the lip would be 0.039 rather than 0.05. Probably good enough.

Acknowledgement

Thanks to Dennis Jewitt for finding an error in the washer thickness plus reminding me that metric drills can fill in gaps in the drill sizes.

Thanks to “Toolmaker51” of homemadetools.net for his insights on annealing aluminum.

Thanks to Kent (www.kustomsbykent.com) for his insights on the safe limits of these rivets.

I welcome your comments and questions.

If you wish to be contacted each time I publish an article, email me with just "Subscribe" in the subject line. If you are on this list and have had enough, email me "Unsubscribe" in the subject line.

Rick Sparber
Rgsparber.ha@gmail.com
Rick.Sparber.org