Drill, Tap, and Counterbore
Version 3

By R. G. Sparber

Copyleft protects this document.[1]

This article deals with drilling, tapping, and counterboring holes for someone new to the hobby of metalworking.

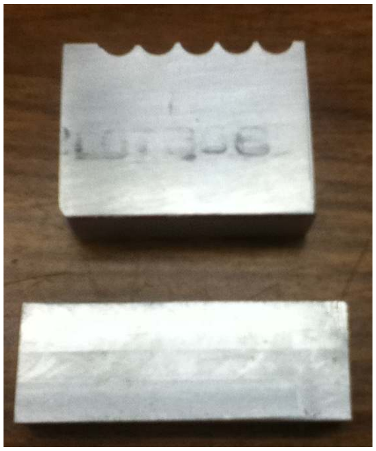
Terminology

Here are a few terms I will be using:

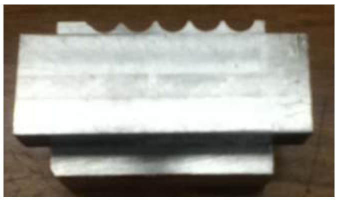
- Tap - a tool that cuts threads into the inside surface of a hole
- Tap hole - a hole with a diameter that will accommodate a given tap
- Clearance hole - a hole with a diameter that will pass a bolt with a given diameter
- Counterbore - a hole of limited depth that will accept the head of a bolt

The Basic Idea

Installing a screw into one or more pieces of metal requires a number of holes to be drilled. These holes must be concentric or the screw will bind up. There are two ways to achieve this alignment. The first is to use a high degree of precision during drilling. Often the location of the screw is not that critical so needing this high degree of precision is best avoided. That leads me to the second method - match drilling. This procedure gives us concentric holes without requiring precision. In most cases, this is the best approach and will be presented here.



I want to attach these two pieces of aluminum together with screws.



After a great deal of thought, I have set the desired arrangement.

I will use what are called Socket Head Cap Screws (SHCS). You can barely make one out at the bottom of this picture. The screw has a hexagonal hole in the top to drive it with an Allen Key.

Here are the tools I will use. On the far left is a 3 flute tap. It will cut a 10-24 thread. The “10” refers to the diameter of the screw that will fit this hole and the “24” is the number of threads per inch.

To its right is my #9 drill which will cut a hole 0.196" in diameter. With it I will form my clearance hole.

To its right is my #25 drill which will cut my tap hole with a diameter of 0.149".

At the far right is a 4 flute end mill that is 11/32" in diameter and will be used to cut my counterbore hole. The tap and clearance holes are specified to work with a 10-24 thread. A given tap has an ideal tap hole diameter. Sometimes this diameter is etched into the tap. Otherwise it can be found in various books like Machinery's Handbook. The clearance hole diameter is often specified as a range of values depending on the desired fit. I have chosen a close fit clearance hole. The end mill is not a perfect fit for my counterbore hole but is the best I could find in my set.

Using layout fluid, I have marked the location of my first screw. Note that both pieces of metal are clamped into position. My spud helps me set the center of rotation of the drill accurately on the cross lines. The stack of metal rests on a piece of scrap Medium Density Fiberboard (MDF) to protect my table.

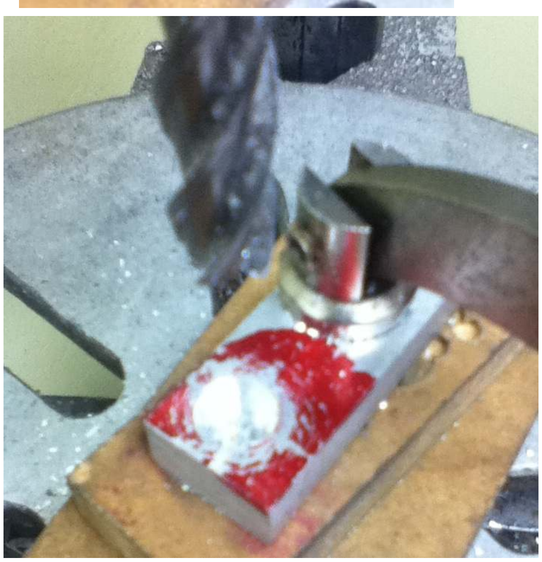
Do not attempt to hold the metal with your fingers while drilling. At best the drill will grab and spin the part with minor damage to the hole. At worst you lose a finger.

I am using a centering drill to make a small cone shaped hole in the metal. This tool has a small cutting diameter but a large body diameter. There is much less tendency for deflection of the drill with this tool. If I used a small diameter drill, it is likely it would bend as I started the hole and when it finally did start to cut, would not be at the cross lines.

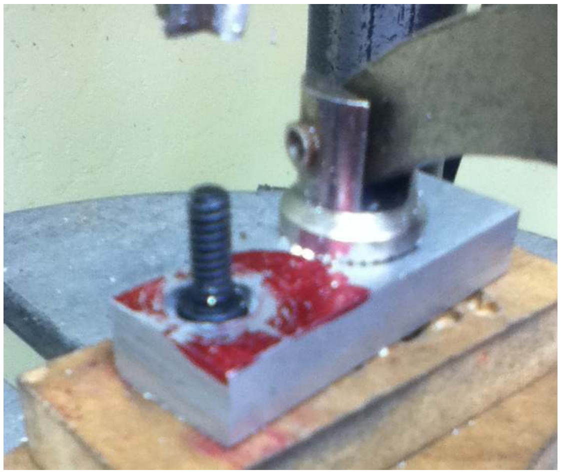
I then used my tap drill and went through both pieces at the same time. This is what match drilling is all about. The resulting hole must perfectly align in both pieces because they were cut in the same pass. The parts have been unclamped because the remaining steps are done separately on each part. Before proceeding, I deburred both holes.

I now recut the top plate with a clearance and counterbore hole. The first step is to align the hole with the center of rotation of the drill chuck. I do this by reinstalling my tap drill and feeding it into the hole. Then I clamp down on the part. I have added a second piece of MDF to raise it up a little because I have limited depth range on my drill press.

With the part aligned on the tap drill, I switch to my clearance drill and go all the way through this top piece.



Then I switch to my end mill to cut the counterbore hole.



I am using the screw's head to gage the depth of the counterbore hole. ***This is done with the machine off. The end mill removes fingers far better than it removes metal.***

The counterbore hole is deep enough when the top of the SHCS is level with the top surface of the plate.

To recap, I aligned the part with the tap drill which insures that the existing hole is centered at the center of rotation of the drill chuck. Then I drill the clearance hole which will be centered. Running the end mill without disturbing the part's position insures that the counterbored hole will also be centered.

The top part is now done.

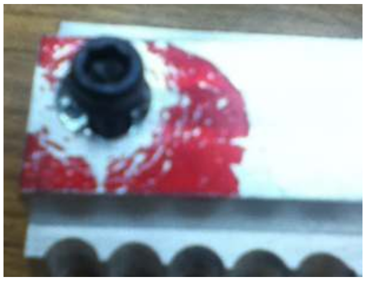
Next I put down my bottom plate. Again I use my tap drill to align the hole with the drill chuck's center of rotation before clamping it.

This next step takes a lot of care to do right. I chuck up my spiral tap and position it above the hole. Power is applied and I wait for full speed to be reached. Then I cut power and plunge the tap into the hole.

A spiral tap is designed for this operation. A hand tap will likely snap off. It is also essential that the spiral tap be sharp to minimize binding. A bit of cutting fluid, like WD-40 also helps.

Terrible picture but I hope you can make out that the tap is stuck in the hole. I will use a tap handle to turn it all the way in and then remove it.

After the tap has been removed, I used a countersink mounted in a brace and bit to deburr the hole. One or two turns does the trick and leaves a very smooth little chamfer.

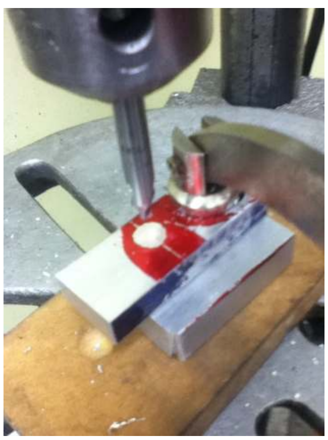


The screw can now be installed. Note that screws are not for precision alignment of the parts but simply to clamp them together.

After further consideration, I decide I wanted a second screw. Its position relative to the first screw is not important but I certainly want to be able to install both screws without binding.

I fully install the first screw. I know this sounds trivial, but *if the second hole is drilled with the first screw in place, I am guaranteed that both screws will fit perfectly.*

So here I have the screwed together parts back on my drill press with a new set of cross lines scribed. Again I used my spud to locate the center of rotation at the intersection of my scribed lines.



I first use my center drill to cut a small cone.

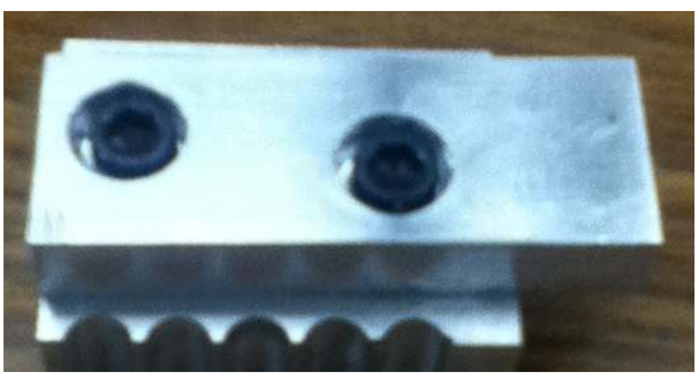
I drill my tap hole all the way through both parts.

I then separated and deburred the two part. After mounting the tap drill again, I use it to align the top part.

I next drill the clearance hole.

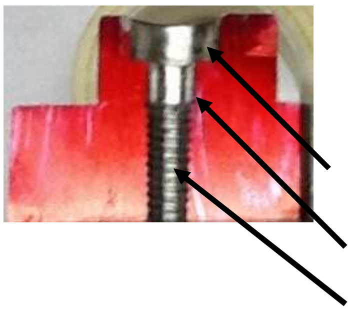
And finally, I use the end mill to cut the counterbore hole. I then remove the part and deburr it.

Next up is the tapping of the bottom plate. I could use the tap drill to set alignment but the tap works fine. The drill press is brought up to speed, then power is cut as I drive the tap into the tap hole.



Once deburred, I installed both screws which have to be a perfect fit due to the method used.

Here is a close up of the screw.



I sawed across one of the holes and used layout dye to improve contrast.

Counterbore hole

Clearance hole

Threaded section

Reader's Comments

As hoped for, I received a number of valuable comments from readers:

- Russ from the Valleymetal Yahoo group pointed out that although an end mill can cut the counterbore, a better job can be achieved with a counterboring tool. One example is:

  http://www.use-enco.com/CGI/INSRIT?PMAKA=368-0106&PMPXNO=945110&PARTPG=INLMK3

- Ron from gingery_machines Yahoo group pointed out that holding an endmill in a drill chuck is not such a good idea. It can damage the chuck because the endmill is harder than the chuck's jaws. If the endmill spins, it can score the jaws. Had I done the drilling and counterboring on my mill, I would have put the endmill in a proper collet and avoided this risk.

- JRW from valleymetal Yahoo group made two important points. The first relates to stamping parts after match drilling to insure the right holes line up. An alternate approach is to place the holes such that matching the wrong holes causes the parts to be way out of alignment. His second point is that I really didn't use the best tool for cutting my starter cone. A center drill is really design to cut a cone in stock to be mounted on a lathe. A spotting drill is the proper tool. I have to admit that I did know better but could not find the proper size.

  Here you see two of my spotting drills on the left compared to a center drill on the right.

  The main problem with using the center drill is if you drill too deep. That little point can break off. In my feeble defense, I did only drill in enough to cut the cone so used only the tip.

- Dave from gingery_machines points out that all of the drilling steps for a hole can be done without disturbing the stack of plates. First drill the tap hole through. Then set up a means of limiting the depth of the clearance hole. This means can be as simple as a block of wood with a hole drilled in it with the needed thickness. Then counterbore with a similar depth stop. And finally, run the tap through the clearance hole and through the tap hole.

I welcome your comments and questions. All of us are smarter than any one of us.

Rick Sparber
Rgsparber@aol.com