Drilling holes in thin sheetmetal

Have you ever tried to drill holes in thin sheet metal, using standard drills, and ending up with a ragged triangular shaped hole, not anywhere near the diameter you wanted, and perhaps requiring a trip the medicine cabinet for a band aid?

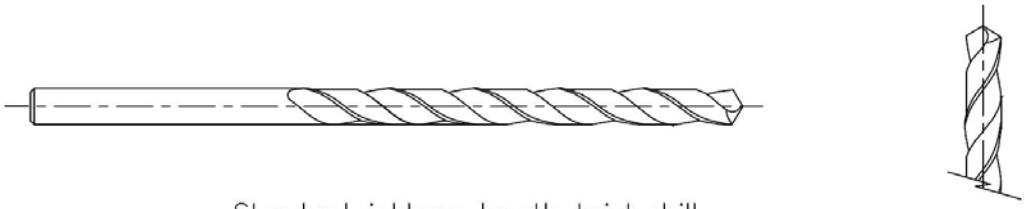
Compare the two holes above, in a piece of .050” thk. aluminum. Both holes were drilled with a 1/2” dia. drill at about 150 rpm, with a wood support on the backside. The hole on the left was drilled with a standard twist drill, the one on the right with a modified tip. Neither hole has been de-burred.

Not everyone has the good fortune to have a punch set that will punch thin sheet metal in all the different sizes required for our electronics chassis or enclosures. These instructions will enable you to grind a standard twist drill with a point that will enable you to drill thin sheet metal. You can even use these drill to drill holes in the infamous “Altoids” tins. I was shown how to grind these drills many years ago, as an apprentice in a small custom sheet metal shop. This technique applies to 1/8” diameter to 5/8” diameter drills. When I was younger, I could do drills a little smaller than that, but those days are gone now. You will need a bench grinder with a medium to fine wheel, and a dressing stone to give the wheel a sharp corner for the smaller sizes. I don’t do every diameter drill in this style, because once you have a perfectly round hole it is easier to open it slightly with a standard drill or tapered reamer, rather than grind up a new diameter. I keep a range for the controls and switches I use a lot, and diameters that don’t have things mounted in them, that are exposed to view, like speaker holes, etc. I have used this technique for materials from Altoids tins to 1/8” aluminum panels.

I use a inexpensive drill press to use these drills, and it is recommended, but not required. A hand drill can be used with care.

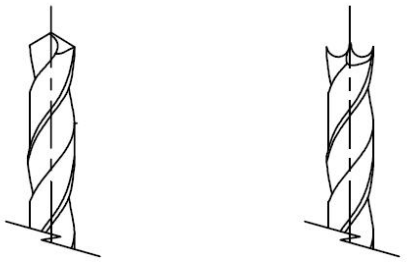
It is assumed that the user knows how to safely use hand tools and will apply the appropriate safety equipment, and practices.

Here are a few of my sheet metal drills. These range from about 3/16” to 5/8” in diameter. If you only drill aluminum, they will stay sharp for quite some time. Use slow speeds on steel, as the tips that cut the diameter have a small cross section and have little mass to dissipate heat. Always use oil when cutting and they will last quite a while before they have to be touched up again, for sharpening. The diagrams and photos will show you how to grind the tip.

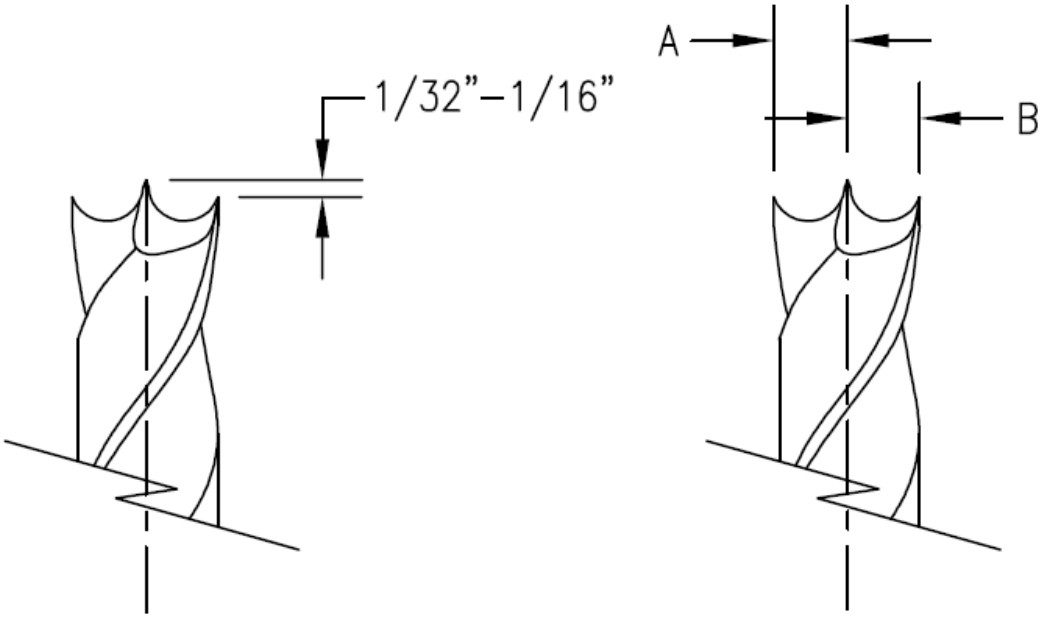


Standard jobbers length twist drill

Here is an illustration of a standard drill bit you can expect to find at any hardware store or the bottom of your tool box. Look at garage sales for drills that are dull and unusable anymore. You are liable to find a box of miscellaneous diameters very inexpensively.



Above, and on the right, is an illustration of the modification we are going to grind on the end.



The above illustration is what you want to achieve, using the edge of the grinding wheel, you want to grind two radii on the tip of the drill as shown. A and B dimensions must be equal, if you are trying to hold the diameter of your hole to the size of the drill. The hole diameter will end up double the distance of either A or B’s distance, whichever is greater, to the center of the drill. So, keeping them equal is of importance if you are trying to dill a hole an exact diameter. Most times, within .005” in diameter is acceptable, and east to achieve.

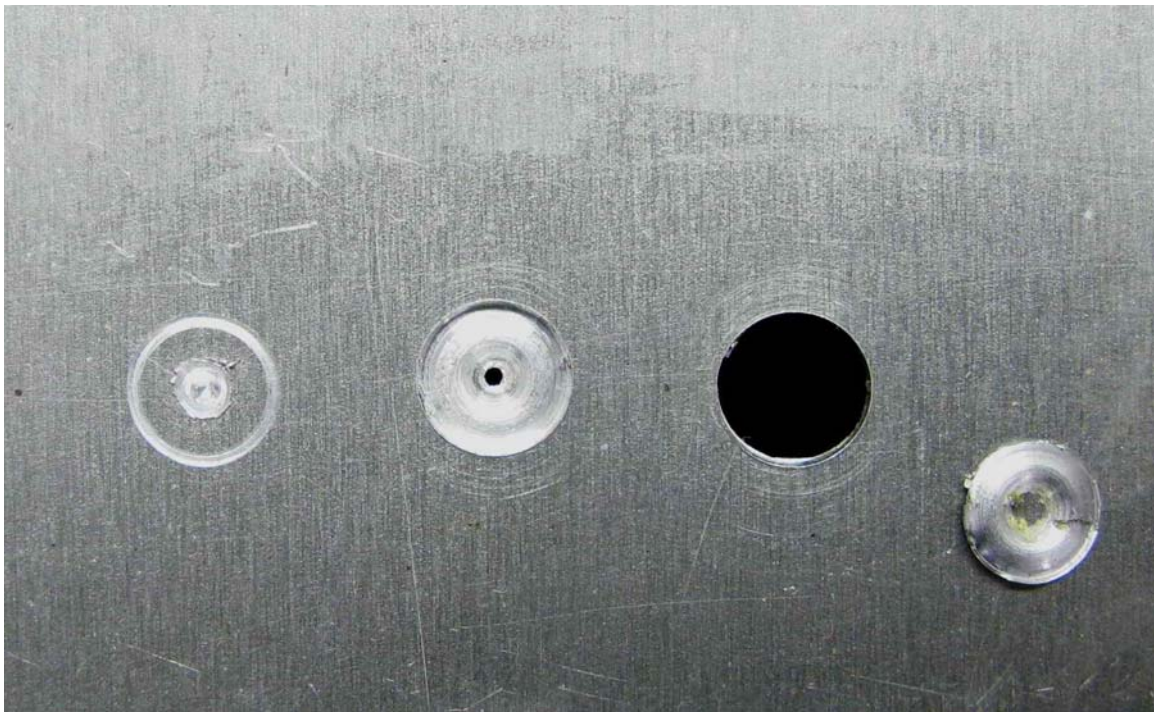
It is very important that the 1/32”-1/16” dimension be maintained. This gives you a point, at the end of the drill, to place in your center punch mark to start. If the point is even with the outer tips of the drill, you cannot pick up you mark, and the drill will walk around the surface, as you try to drill the hole. The scallop you see on the right side of the drill, in these views, is the relief angle that must be present on both flutes. Both flutes of the drill are ground the same. The illustration above only shows the view from one side.

Grinding the tip

Use the appropriate eye protection when using a bench grinder.

This is how to prepare a 1/4” diameter drill. After dressing the wheel so that the wheel does not have a large corner radius, grind the flute to the left of the centerline of the drill as shown, and try to match the earlier illustrations. Rotate the drill 180°, and grind the other side exactly as the first. Note the end I’m holding it tilted down to give you the relief angle we spoke of earlier. The exact angle in not important, 15°-45°, but is necessary so the heel of the relief does not drag on the material being cut. This takes practice, and you have a lot of flute length to play with.

Simple pictures are worth many words. The above pictures are what we are looking for in a finished drill.



Test drill a hole. At the left is the hole just started and starting to drill some of the diameter. The center is about half way through the .050” thickness, with the point already through the material. The right shows the finished hole. At the lower right is the slug that is produced as the drill passes through the back side of the material being drilled. If the diameter is oversize and is not close enough to your desired diameter, this indicates that the center tip is not exactly centered on the axis of the drill. Simply re-grind and test again.

This process can be applied to the common diameters you will encounter doing small chassis work. Larger diameters are best suited using chassis punches or hole saws.

Ken – wa4mnt
www.qrpbuilder.com