

Drill Press



Photo by Curt Newton

A Three-Drill Press in the Laboratory

- [Labeled Photograph](#)
- [Description](#)
- [Using a Center Finder](#)
- [Drilling a Hole](#)
- [Deburring a Hole](#)
- [Reaming a Hole](#)
- [Thread Standards](#)
- [Tapping a Hole](#)

Description

A drill press is preferable to a hand drill when the location and orientation of the hole must be controlled accurately. A drill press is composed of a base that supports a column, the column in turn supports a table. Work can be supported on the table with a vise or hold down clamps, or the table can be swiveled out of the way to allow tall work to be supported directly on the base. Height of the table can be adjusted with a table lift crank than locked in place with a table lock. The column also supports a head containing a motor. The motor turns the spindle at a speed controlled by a variable speed control dial. The spindle holds a drill chuck to hold the cutting tools (drill bits, center drills, deburring tools, etc.).

The quill is moved up and down with a lever on the side as shown here.

Using a Center Finder

A center finder is useful for setting the spindle of a drill press or mill accurately over a known point. A center finder is made of two separate pieces spring loaded together.

The center finder is used by installing it in the chuck, placing the pointed end into a centerpunch mark, and pressing down *lightly* on the quill. At first, the two parts will probably not be concentric.

The bed of the drill or mill can then be adjusted to until the two halves are concentric. Check for concentricity by running your fingers up and down the center finder. You should detect no steps. You should be able to locate the desired position to within .001 inches using this technique. Now the hole is ready to be drilled.

Drilling a Hole

First, a center drill should be used. A center drill has a thick shaft and very short flutes. It is therefore very stiff and won't walk as the hole is getting started. It doesn't cut as easily as a drill bit, so you should use cutting fluid.

Now the hole can be drilled. If the hole is large, its a good idea to drill a smaller pilot hole before drilling the final one. Your hole will be more accurately positioned, rounder, and the bits will last longer. If the hole is deeper than it is wide. use coolant and back off occasionally to clear the chips. As you step up in drill size, you will need to reduce the spindle speed. If drilling a through hole, ensure that the bit will not drill the table after moving through your work. To set a desired depth of hole, there is a depth stop on the quill.

Deburring a Hole

Usually, the top edge of the hole will be fairly clean, but the bottom edge will have substantial burrs. To remove them, insert a deburring tool into the hole and run the tool about the edge of the hole with moderate pressure.

Reaming a Hole

A drilled hole will be accurate to about two thousandths of an inch in diameter. If greater precision is required for slip fits or interference fits, a reamer must be used. The straight flutes of a reamer cannot drill a hole. You must drill a hole slightly undersize to start. Be sure to drive the reamer down with a constant, slow speed.

Thread Standards

The threads cut by taps and dies conform to a standard for the shape of the threads. Often it is the *American National Standard* in National Coarse (NC) or National Fine (NF). NC has fewer threads per inch than NF. NC is most common while NF is favored in precision assemblies. Also in common use is the *Unified Thread System* with UNC (coarse) and UNF (fine). The only difference between National and Unified threads is the shape of the root and crest. The threads in the two different standards will mate. Fasteners are designated by their diameter, number of threads per inch, and shape. For example, 1/4-20NC means 1/4 inch diameter, 20 threads per inch, and National Standard. Threads smaller than 1/4 inch are designated by number from 0 (smallest) to 12 (largest). To convert number to diameter multiply by 0.013 and add 0.060.

Tapping a Hole

If you want to cut threads in a hole, use a tap. The picture below shows what a tap looks like. It has cutting edges to cut the threads and straight flutes to allow chips to be expelled. Note that the end is tapered slightly to help the tap get started. Taps and dies are hard and brittle so you should be careful working with them (try not to drop them or force them into a hole when stuck). Be sure that the hole you drilled is the correct size for the tap you're using or it may break inside your part. Refer to a chart or machinist's handbook. The die in the picture is for cutting external threads on a shaft.

It's best to use the drill press to help maintain alignment as you tap the hole. First, use the centerfinder to place the spindle directly above the hole. Then install a tapered guide into the chuck. Put the tap in place and apply moderate pressure with the quill as you turn the tap. It's good practice to back the tap up a bit for every quarter turn of thread you cut.
