FOCUS ON: Precision Drill Presses Small Hole Drilling

What is small hole drilling? How small is small?
Why is drill runout important?
What else should I know about?

The Servo Products Company line of precision sensitive drill presses are suited for small hole drilling in the range of 0.004 inch (0.1 mm) to 1/8 inch (3.2 mm). Cutting tools up to 1/4 inch (6.35 mm) can be held by the WW collet type of machines. The size range from 1/8 inch (3.2 mm) to 1/4 inch (6.35 mm) is for work only in most non-ferrous metals and plastics. For purposes of this discussion, we will define small hole drilling as covering the range from 0.004 inch (0.1 mm) to 1/16 inch (1.59 mm).

One of the most important factors involved in small hole drilling is the ability of the cutting tool to accomplish "work" at a predetermined rate. This ability is considerably reduced with small cutting tools over their larger counterparts due to the delicate nature of their construction and the problems encountered with uniform hand feeding rates.

Drill Runout

An additional factor when using small cutting tools is the percentage of "runout" relative to their diameter. As an example, if a 0.500" (12.7 mm) diameter cutting tool has a Total Indicated Runout (TIR) of 5%, this would amount to 0.025 inch (0.64 mm). This amount of runout would be readily noticeable to the operator. However, when the same percentage of TIR is applied to a diameter of 0.005 inch (0.127 mm), this results in a runout of only 0.00025 inch (0.00635 mm) which would not be readily observed by the operator. From the above comparison, it is apparent that while the amount of TIR is relatively small as an absolute amount for the 0.005 inch (0.127 mm), it is just as damaging to the finished hole size and to potential drill breakage as the TIR is for the 0.500 inch (12.7 mm) diameter cutting tool.

The above comparison shows the importance of maintaining the TIR to as low a percentage of the tool diameter as practical. This is the reason the collet system of chucking is preferred over the three-jaw chuck since the TIR for the collet system is approximately 1/5 to 1/3 the value of the three-jaw chuck. Generally speaking, the TIR of the collet system will run within 0.0002 inch (0.0051 mm) to 0.0003 inch (0.0076 mm). The TIR for the three-jaw chuck will run within 0.0005 inch (0.027 mm) to 0.0015 inch (0.0381 mm). Additional factors to minimize tool runout include making the tool overhang (length projecting from collet or chuck) as short as possible, as well as using the shortest possible tool.

Cutting Tool Feeds and Speeds

The spindle’s revolutions per minute (rpm) is important when drilling small holes. For the more exotic **ferrous metals** covering all of the high strength steels including stainless steel, the best rule to follow is to start at a low rpm such as 1,000 and gradually increase the speed as a "feel" is developed for the proper feed rate to maintain a continuous chip structure. When the drill starts to enter the work, it is extremely important to maintain the feed pressure in a sufficient amount to immediately start cutting while at the same time not overloading the drill, which would cause breakage. This technique will prevent "work hardening" on the surface of the material to be drilled during the early entry stages of the drill point. A typical speed range for these materials would be between 1,000 and 5,000 rpm. The rpm for **non-ferrous metals** can generally be increased approximately three times over the ones outlined above. Some materials such as hard brass, etc. would be required to start at a slower speed and slowly increase the rpm until the proper chip is produced. (Continued)

Cutting Tool Feeds and Speeds (continued)

The drilling of **plastics** and printed circuit boards can generally be accomplished in the higher ranges of speeds between 10,000 and 20,000 rpm. When using diamond drills in hard materials such as **glass, ceramics, and sapphire**, for example, rpm's should be in the 2,000 to 5,000 region. It is most important that the diamond drill is permitted to cut the work without being forced and the rate of "pecking" is in the 1-3 second time interval.

Slower feed rates and higher drill speed can help minimize buckling and point walking, but if the material is abrasive and/or subject to work hardening, these conditions can increase tool wear and may cause breakage. Tune the spindle speed and feed rate to reduce harmonics, which can induce vibration and chatter. Also make sure the part is held rigidly.

Maximum depths in drilling small holes from 0.004" (0.10 mm) diameter range up to 20 or more times the diameter. Deep hole drilling of small holes generally requires slower speeds, and is usually accomplished by gradually increasing the hole diameter and depth, using the most rigid drills possible. Peck drilling (frequent tool withdrawal) is generally required to clear chips and prevent packing, and to apply lubricant.

Material Considerations

As the hole gets smaller, the material being drilled acts less homogeneous and less machinable. Materials with inclusions, microscopic voids, carbide or alloy segregates, and the grain structure itself can cause problems when using extremely small drills. The hole should start on a flat, unblemished surface perpendicular to the axis of the drill or at a pre-drilled center or pilot hole. When a drill's chisel edge first contacts the drilling surface, irregularities on that surface may immediately deflect the drill, causing it to break. Also, make sure that the surface was not work-hardened by a prior operation.

Drills

The type and quality of the drill bit itself is important. Center, pivot, spade, and straight shank drills are among the types used for small hole drilling. Small drills are designed to compensate for loss of rigidity. The webs or center sections of small diameter drills are thicker than those of larger drills because it is not practical to manufacture them with the same percentage of web. This increases the rigidity of the drills, but it also decreases the length of the cutting lips. Also, thicker webs at the points increase the end pressures required to force the drills into the workpieces. Tool materials include high-speed steels, cobalt (M42), carbide, and diamond. You may need to test various types to find the correct drill for your application. Center drills are the most rigid. Inspect the drills; even new tools are not necessarily made right, and even small variations in tool geometry, such as web thickness or off-center points, make the drilling process and tool life unpredictable. The annual "Buyers Guide" issue of *Cutting Tool Engineering* magazine lists manufacturers of miniature drills. (www.ctemag.com or 847-498-9100)

Cutting fluids

For wet cutting, the preferred cutting fluids are a water soluble coolant or thin oil. Heavier viscosity fluids, such as cutting oil, tend to work against chip evacuation on small diameter holes. They do, however, become necessary on materials that require more lubricity during the cut.

> **NOTE** *This bulletin was compiled from a variety of industry sources and is provided solely for customer information in regards to small hole drilling guidelines. Servo Products Company can in no way assume any responsibility for damage.*

For Additional Information

Contact your Servo dealer or Servo Regional Sales Manager or
Servo Products Company Technical Support:
Phone: 800-521-7359 or 440-942-9999 or FAX 440-942-9100
Website: www.servoproductsco.com