The Use and Care of Twist Drills

Twist drills are designed to create new holes. Drill geometry is designed to spread the cutting load from the center of the point outward. Drills are not designed to enlarge pre-existing holes. When force is exerted on the outer cutting lips (the weakest part of the drill), the tool will chip, and ultimately fail.

Many drill users insist on drilling pilot holes prior to drilling the final desired hole size. Split point drills eliminate the need for pilot hole drilling. If the material does require a pilot hole, it is important to drill the smallest possible pilot hole so that only the drill's chisel point engages the material.

Chipped outer cutting lips often occur when the twist drill is used to enlarge a hole. A telltale sign of this type of abuse is that little or no wear appears at the chisel edge. This is most common with Silver & Deming drills and large diameter drills.

Common causes of chipped lips

- Drill used to enlarge an existing hole
- Often results from drilling a pilot hole too large
- Pilot holes should never be drilled prior to using a split point twist drill

Chassis reamers (or car reamers) enlarge holes. Reamers cut along their outside edges, spreading the cutting loads along the broad edges of the cutting tool.

Speeds and Feeds

The speed of a drill is usually measured in terms of the rate which the outside or periphery of the tool moves in relation to the work being drilled. This is commonly referred to as "Surface Feet Per Minute" in the machine shop and manufacturing environment. The formula is as follows:

SFM=.26 X RPM X Drill Dia. In Inches

In general terms, increase in speed result in fewer holes before dulling occurs. The challenge of choosing the correct speed requires a balance between the best possible production rates vs. best tool life. The best speed for operating a drill depends on:

- Composition and hardness of material
- Depth of hole
- Efficiency of cutting fluid
- Condition of drilling machine
- Difficulty of set-up

The rapid wearing away of the extreme outer corners of the cutting edges indicates that the speed is too high.

Feed rates are governed by the diameter of the tool and the material being drilled. In general, larger diameter drills must be run slower than smaller diameter drills. *In the machine shop environment, a drill split up the middle (of the web) is evidence of too much feed.*

The most common errors in the operation of small drills are over-speeding and under-feeding. Feed should be based on the thickness of chip. Speed should then be adjusted accordingly.

Successful drilling depends on feeds that will actually produce CHIPS and not POWDER.

Lubrication

Cutting fluids and waxes are of great advantage in most metal cutting operations.

1. Lubrication-the purpose is to lubricate the contact surfaces between the tool and the work to reduce friction and heat. Lubrication of the chip provides better chip disposal, and reduced heat. Lubrication also prevents the chip from welding to the tool, which can cause “build up edge” and shorten cutting tool life dramatically.
2. Cooling-in production work, cutting fluids provide the primary role of cooling of the drill. Cutting fluids should be directed at the point of contact for best results.

Rigidity

The work-piece must be held securely for efficient drilling. It should also be supported so that it does not bend due to drilling pressure. This is difficult when the material is thin and frail, but it should be remembered that bending will cause oversize and out-of-line holes, as well as excessive drill breakage.

Drilling Tough and Hard Materials

It is difficult to drill holes in hard and tough materials which are commonly used.
See that the work is well supported directly under the drill point, and held rigidly in place. Use a powerful drilling machine, preferably corded. Use a short drill, preferably a screw machine or mechanics length drill. Use medium to heavy feed pressure and reduce drilling speeds.

P.O. Box 368, Rockville Centre, NY 11571-0368
Phone 516.536.8200 ~ Fax 516.536.8186
www.championcuttingtool.com

Stainless Steel

More difficult to drill than carbon steels. Some are free-machining, but the 18-8 types are very difficult due to work-hardening properties. *First rule is to keep drills continuously cutting.* If tool edges are permitted to idle and rub on the work, the surface will be work-hardened and it will be impossible to restart the cut. For the same reason, it is advisable to use fairly heavy feeds, so that the tool edges will get under the work-hardened surface left by the preceding edges.

Armor Plate

Very difficult to drill due to extreme hardness and toughness. Tendency to work-harden, and is often flame cut which produces hard spots. Make sure cutting tool cuts continuously to avoid work-hardening, use heavy feed pressure. Drills must be kept sharp. Use a large volume of cutting fluid to reduce heat.

Molded Plastics

Very popular, and produced from a variety of materials known as resins, phenolics, acetates, polystyrene, acrylics, etc. The ejection of chips is the biggest problem, and overheating can cause destruction to work-piece and cutting tool. Compressed air is often used to keep work-piece and tool cool.

Sheet Metal

Drilling of sheet metal presents special problems. The main problem is drill breakage, particularly in small diameter holes. Because it is most often drilled using a portable hand-held drill, no rigid support is provided, and there is inconsistent feed pressure. This produces a distinct shock at the point during break-through.

Use the shortest available drill for drilling sheet metal.(Screw machine drill or “stubby”, or mechanics length drill).

Deep Hole Drilling

Speeds should be reduced from 10 to 40 percent when drilling deep holes because of the problems associated with chip disposal and greater heat build-up.

P.O. Box 368, Rockville Centre, NY 11571-0368
Phone 516.536.8200 ~ Fax 516.536.8186
www.championcuttingtool.com