DRILL SPEED AND FEED RECOMMENDATIONS

Material		Speed SFM	Feed (I.P.R.) 1/16"	1/8"	1/4"	1/2"	3/4"
Aluminum / Aluminum Alloys		300 - 600	.0008"	.003"	.007"	.012"	.015"
Aluminum Alloyed Si > 10%		150 - 400	.0008"	.002"	.006"	.010"	.012"
Cast Irons	Soft	200 - 300	.001"	.003"	.005"	.010"	.012"
	Medium	125 - 225	.001"	.003"	.005"	.008"	.010"
	Malleable	65 - 200	.0005"	.002"	.004"	.007"	.010"
	Brass	200 - 300	.0007"	.002"	.003"	.004"	.006"
	Bronze	150 - 250	.0007"	.002"	.003"	.004"	.006"
Coppers / Copper Alloys		150 - 300	.001"	.003"	.006"	.010"	.012"
Magnesium		300 - 600	.001"	.003"	.007"	.012"	.015"
Nickel Alloys		75 - 200	.001"	.003"	.005"	.009"	.012"
Stainless Steels	Free Machining	100 - 150	.001"	.003"	.005"	.008"	.012"
	Work Hardening	50 - 100	.0005"	.002"	.004"	.006"	.010"
Steels	Low Carbon	150 - 300	.001"	.002"	.004"	.007"	.012"
	Medium Carbon	100 - 200	.001"	.002"	.003"	.006"	.010"
	High Tensile (35 - 40Rc)	75 - 150	.001"	.002"	.003"	.004"	.005"
	High Tensile (40 - 45Rc)	50 - 100	.0007"	.001"	.002"	.003"	.004"
	High Tensile (45Rc +)	25 - 75	.0005"	.0007"	.001"	.002"	.003"
Tool Steels		40 - 100	.001"	.0015"	.003"	.005"	.008"
Titanium	Soft	80 - 125	.001"	.002"	.004"	.006"	.010"
	Hard	40 - 100	.0007"	.001"	.002"	.005"	.008"

OPERATING PARAMETERS

To achieve optimal performance, attention must be paid to the following:

1. Machine utilizing tooling must have necessary rigidity to minimize spindle deflection and sufficient horsepower to perform at recommended speeds and feeds.
2. Make sure holders and collets give good concentricity between tool and machine spindle.
3. Rigidly clamp and support workpiece to minimize deflection.
4. Use as short a drill as the application will permit to give maximum tool rigidity.
5. Use coolant, as recommended, to improve tool life. Direct the flow of coolant to the cutting edges. Insufficient or poorly directed coolant can result in poor tool life.
6. Use the correct speeds and feeds to suit the application and material being machined.
7. Resharpen or replace drills at first indication of cutting lip dulling or corner of cutting lip rounding.

Speeds and Feeds for Deep-Hole Drilling

Holes which must be drilled three diameters deep or more fall into the "deep hole" drilling class and some adjustment of feeds and speeds is necessary.

The deeper the hole, the greater the tendency there is for chips to pack and clog the flutes of the drill. This increases the amount of heat generated and prevents the coolant from conducting the heat away from the point. A buildup of heat at the point will eventually result in premature failure.

Peck drilling, or the practice of drilling a short distance, then withdrawing the drill, will often reduce the chip packing. The deeper the hole, the more frequent the drill must be retracted to be effective.

A reduction in speed and feed to reduce the amount of heat generated is generally required in most deep-hole applications where coolant cannot be effectively applied.