Speeds and Feeds

Improper speeds and feeds can drastically affect the performance of any cutting tool, use of the appropriate speeds and feeds is even more critical for multi-diameter tooling

In general, speeds are specified in metres per minute or revolutions per minute (rpm) while feeds are related in millimetres per minute or millimetres per revolution. Speeds for multi-diameter tooling should be selected by the mean diameter. This diameter can be determined by adding together the largest and smallest cutting diameters of the tool and dividing the sum by two. Feed rates for multi-diameter tools should be in conjunction with the smallest cutting diameter. Of course, there are applications in which one of the extreme diameters is functioning to a greater degree than the other and should govern the speed and feed selection.

A guide to speeds and feeds is illustrated on page 26. These are only suggested starting ranges and may have to be refined due to operational variations. Other factors influencing a deviation from this guide would be:
Deep hole drilling - a reduction in speed and feed in proportion to depth of cut should be considered (usually on a trial and error basis).

Core drill - feed rates can generally be increased from 50% to 100%.
Reaming - reduce speed to approximately 60% to 70% of the speed specified for drilling and feed at a rate of 0.05mm - 0.076mm per cutting edge.

Web Styles

The web or core section of fluted cutting tools, particularly the drill type, vary with the material and operation it is applied to, and will increase in area as the degree of machining difficulty increases.

Core or reamer type tools usually have straight webs ranging from 40% to 70% of the tool diameter.

Margins

Multi-margins on drill type tools greatly increase support area in guide bushings or part cavity. This added support will generally improve overall cavity quality. (see page 23)

The basic criteria for applying narrower than standard cylindrical margins to cutting tools is:
- closer control of diameters and finish of machined cavity
- non-ferrous material being removed form pre-existent cavity.

Spirals

A spiral range of 15º to 32ºis adequate for the majority of drilling and core drilling applications when using H.S.S., in general carbide tipped tools have slower spirals. When drilling from the solid to depths over three or four diameters a faster spiral should be considered to facilitate chip removal. For reamers, counterbores, spotfacers, tools which remove less material or cut to shallow depth, a slower spiral is recommended since it has less tendency to hog-in to the work material, and creates a flute geometry more adept to producing true part surfaces (see page 22)