TURNING AND MILLING BISPLATE®

MILLING AND TURNING RECOMMENDATIONS

MILLING

Milling operations can be performed satisfactorily on all BISPLATE® grades; utilisation of cemented carbide tooling is recommended.

In many situations, the milling operation entails the dressing of a flame cut edge, and then subsequent bulk milling of material to the desired surface finish and dimensional tolerance.

Care must be taken to make a first cut sufficiently deep to remove the heat affected zone of the flame cut edge. Cutters must be sufficiently robust to take this heavy loading. In such circumstances it is desirable that, due to the high hardness adjacent to the flame cut surfaces, cutter speeds and feed rates for initial milling should be reduced to 40-50% of the speed normally used when milling plain carbon steel. The importance of adequate preheating prior to flame cutting and slow cooling after cutting to minimise edge hardening is again emphasised. Speed and feed rates may be increased somewhat for subsequent bulk milling to 50-75% of the settings used for plain carbon steel.

MILLING RECOMMENDATIONS

Table 1:

BISPLATE® GRADE	CEMENTED CARBIDE TOOLING GRADE	SURFACE SPEED	FEED/TOOTH
60	GC4030	295m/min	0.25mm
70	GC4030	275m/min	0.25mm
80	GC4030	257m/min	0.25mm
320	GC4030	131m/min	0.25mm
400	GC4030	110m/min	0.25mm
450	GC4030	100m/min	0.25mm
500	GC4030	87m/min	0.25mm

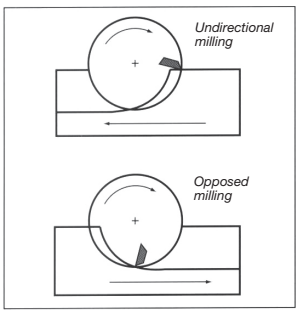
Note: These recommendations are given as a guide only, and are based on stable working conditions. It is suggested a 45 deg. approach angle or a round insert facemill be used. In certain conditions it may be necessary to use negative geometry milling tools. Feed rates are dependant on geometry selected. Eg. PM medium machining (0.1 – 0.28) fz mm/tooth PH heavy machining (0.1 – 0.42) fz mm/tooth.

AVOID VIBRATIONS

Indexable inserts are sensitive to vibrations. These can be avoided or reduced by observing the following.

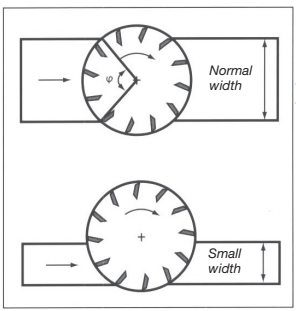
When turning or milling gas cut edges the cutting depth should be at least 2mm to cope with the hardness and unevenness of the edge.

Figure 1:



Showing the direction of milling.

Figure 2:



Showing the eccentricity milling cutter.

OTHER MILLING REQUIREMENTS

- Firm clamping of the workpiece
- Use cutters with the smallest possible gap between the teeth
- Machine stability permitting, unidirectional milling is preferable, see figure 1
- If a large cutter is used for the milling of small areas, place the milling cutter eccentrically to get as many teeth as possible operating, see figure 2
- Avoid, if possible, the use of a universal cutterhead which generally causes weakening of the power transmission and the tool holder

TURNING

All BISPLATE® grades, including those with hardness in excess of 360HB can be turned satisfactorily with carbide tooling, provided spindle speeds and feed rates are reduced from those normally employed when carrying out similar machining operations on plain carbon steel. Reductions of 50-70% in spindle speed and up to 50% in feed rate may be necessary, depending on the hardness of the component being machined. High speed tools are not recommended.

As an example, the following settings have been found to give satisfactory results when turning cylindrical workpieces of 25mm diameter from the various BISPLATE® grades. With increases in stock diameter, spindle speeds will obviously need to be decreased.

TURNING RECOMMENDATIONS

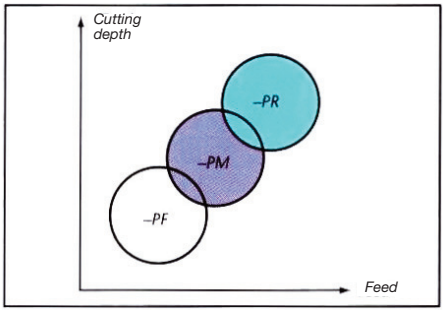
Table 2:

BISPLATE® GRADE	CEMENTED CARBIDE TOOLING GRADE	SURFACE SPEED
60	GC4025	295m/min
70	GC4025	275m/min
80	GC4025	257m/min
320	GC4025	131m/min
400	GC4025	110m/min
450	GC4025	100m/min
500	GC4025	87m/min

For operations under favourable conditions where higher productivity can be obtained GC4015 could be used.
For operations with high toughness requirements and where increased security is needed GC4035 could be used.

Note: These recommendations are given as a guide only. And are based on stable working conditions. The geometry of the inserts used will be dependant on the operation.
Eg. PF for finishing. PM for medium machining. PR for roughing.

Figure 3:



Geometry of Turning Insert: Finishing (F) Medium Machining (M) Roughing (R)

OTHER TURNING REQUIREMENTS

- Firm clamping of the workpiece
- Avoid long overhangs for both workpiece and tool holder
- Use correct tip radius: too large a tip radius, combined with insufficient clamping, causes vibrations
- Small setting angles also can cause vibrations

Figure 4:

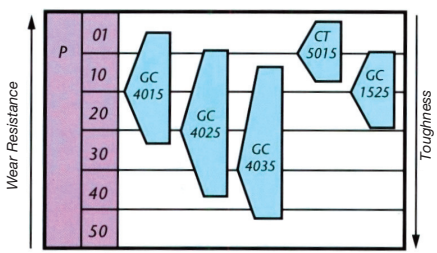


Diagram showing that some coated inserts combine toughness and hardness.

FORMULA FOR THE CALCULATION OF SPEEDS AND FEEDS FOR GENERAL MILLING AND TURNING OPERATIONS.

Formula for calculation of cutting speed:

$$v = \frac{\pi \bullet D \bullet n}{1000} \quad \text{m/min}$$

Formula for calculation of turning speed:

$$n = \frac{v \bullet 1000}{\pi \bullet D} \quad \text{m/min}$$

Formula for calculation of table feed:

$$u = n \bullet Z \bullet Sz \quad \text{m/min}$$

v = cutting speed m/min
D = Diameter in mm of milling cutter or workpiece
Z = Number of cutters
Sz = Feed per cutter, mm
n = Turning speed, rpm
u = Table feed, mm/min

Bisalloy Steels® wish to thank Sandvik Coromant for information pertaining to milling and turning contained in this publication.