Machining Instructions

General

Orkot® materials are readily machinable by conventional machine shop techniques. As a general guide, methods used for brass, aluminium or lignum vitae will apply for Orkot® materials. It is preferable to use tungsten carbide turning tools with cutting speeds of 5.5 metres (19 feet) per second. Orkot® materials must be machined dry without the use of coolant.

Turning

Tungsten carbide tooling of the butt welded type using K20 grade carbide is suitable for most applications. If carbide inserts are used, then aluminium grades with high positive rates give best results e.g. Plansee grade H10T, Sandvik H10A or H13A, Mitsubishi HTI10.

For heavy wall thickness, the internal and external diameters should be machined together to reduce vibration.

No asbestos is used in the manufacturing of Orkot® Hydro and the material is completely non toxic. It is however advisable to use adequate dust extraction when machining. If unavailable, operators should wear dust particle masks.

For small volume work and machining of chamfers, radii and other forms, then high speed steel gives good results, but tool life is shorter than with tungsten carbide.

Cutting Angle for Tools

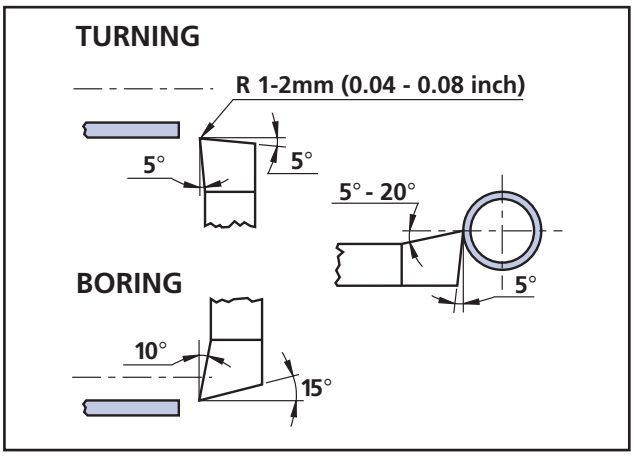

Figure 11: Turning and boring

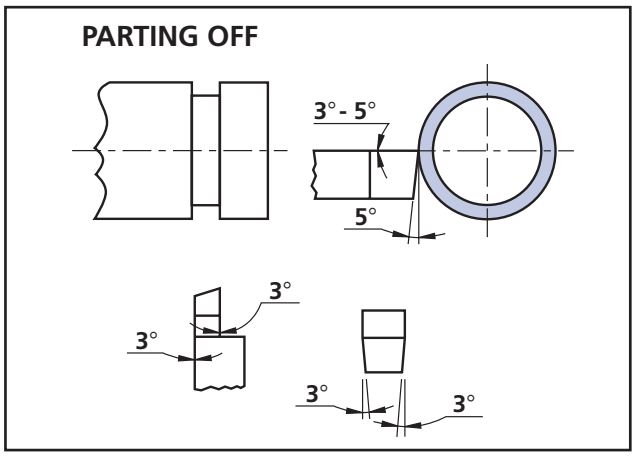

Figure 12: Parting off

Table 14: Speeds in mm

Diameter (mm)	Rpm
0 - 50	2100
50 - 100	1000
100 - 150	700
150 - 200	550
200 - 300	350
300 - 400	250
400 - 500	200
500 - 600	175
600 - 700	150
700 - 800	130
800 - 900	120
900 - 1000	100

Table 15: Speeds in inches

Diameter (inch)	Rpm
0 - 2	2100
2 - 4	1000
4 - 6	700
6 - 8	550
8 - 12	350
12 - 16	250
16 - 20	200
20 - 24	175
24 - 28	150
28 - 32	130
32 - 36	120
36 - 40	100

Table 16: Feed Rates in mm

Type of machining	Roughing	Finishing	Unit
Turning	0.7	0.25	mm/rev
Boring	0.5	0.20	mm/rev
Parting	0.4	0.20	mm/rev

Table 17: Feed Rates in inches

Type of machining	Roughing	Finishing	Unit
Turning	0.028	0.010	inch/rev
Boring	0.020	0.008	inch/rev
Parting	0.016	0.008	inch/rev

Grooving

Orkot® materials can be readily grooved on a lathe, shaping, milling or boring machine with a 90 degree machining head. For most one off applications a lathe is adequate. A sharp high speed steel tool ground to the correct form should be clamped in a long boring bar with a three degree clearance ground on the side of the tool. No top clearance is required.

The chuck may be marked for the correctly spaced number of grooves and each groove shaped in turn. A 0.2 mm (0.008") depth of cut should be used, for long bearings a steady may be required. The machine fast traverse, (with the spindle locked) can often be used. Linear speeds up to 10 m/min or 30 feet/min can be achieved.

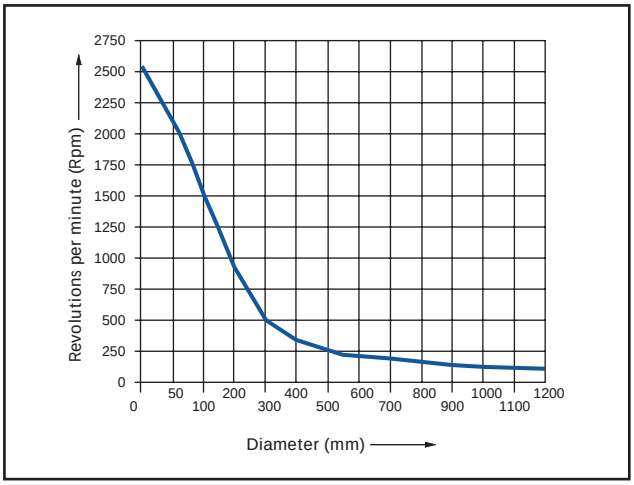

Figure 13: Machining Speeds as a function of Rpm and diameter (mm)

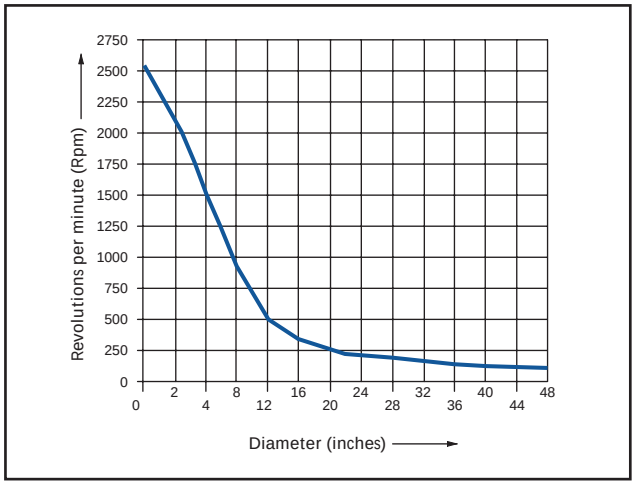

Figure 14: Machining Speeds as a function of Rpm and diameter (inches)

Drilling

Orkot® materials are easily drilled using either conventional high speed steel or carbide tipped drills.

The following speed and feeds are suggested:

Table 18: Speeds and Feeds by Drilling

Drill Diameter mm	Speed Rpm	Feed mm/min
5	1600	300
10	800	400
15	600	400
20	400	400
25	350	400
30	300	400

Table 19: Speeds and Feeds by Drilling

Drill Diameter inch	Speed Rpm	Feed inch/min
0.2	1600	12
0.4	800	16
0.6	600	16
0.8	400	16
1.0	350	16
1.2	300	16

Depth of Cut

Roughing 10mm or 0.4inch
Finishing 3mm or 0.12inch

Smaller cuts may lead to tools rubbing, causing wear which produces excessive heat build up in the finished part.