HINTS AND TIPS FOR MACHINING

Problem	Suggested Remedy
Chatter	Reduce cutting speed. Increase feed rate, sharpen tool, check rigidity of machine or set up, reduce nose radius of lathe tool, check spindle bearings, etc. reduce tool and work overhang.
Poor surface finish	Sharpen tool, increase cutting speed, reduce feed rate, increase nose redius of lathe tool.
Rapid tool wear	Reduce surface speed. Use harder grade of tool material, use cutting fluid.
Tool breakage	Reduce depth of cut, reduce feed, increase cutting speed, use more rigid set up, sharpen tool, check alignment of tool.
Tap breakage	Tap not square, tap jamming on bottom of hole, tapping too far with taper tap, too small a tapping size, swarf jamming tap, use cutting compound.

CERMET, CERAMIC & CARBIDE CUTTING DATA

Tool material	Application
Cermet (Titanium carbide)	Light cutting and finishing of most materials. Max depth of cut approx 2mm. Feed of 0.1-0.2mm per rev. Surface speed 100-300m/min. Cermets will give a high surface finish due to low affinity to other metals, they also have good life due to high hardness,
Ceramic (Aluminium oxide)	Light cutting of hardened steels and cast irons from 55-65 Rockwell "C" (can cut HSS). Modern ceramics are much tougher than early products and can be used successfully in small lathes (Myford size) in good condition, they can also be used on interrupted cuts. Surface speed 80-300m/min. max depth of cut 0.5mm, feed 0.05-0.1mm per rev.
Coated carbide	The coated carbides are the most popular cutting materials used in industry and recommended for the home machinist. They offer the benefit of increased metal removal rate with increased tool life in both turning and milling relative to cost. They suit most steels and can be used for rough turning and finishing at speeds of 80-175m/min and feeds of 0.2-0.4mm/rev.
Un-coated carbide	Generally used for milling applications. An edge preparation gives the carbide a longer life in milling operations. Available in both steel and cast iron grades. Cutting speeds 80-130m/min.

If you consider the followiing factors when turning, you may improve your results.

- Forgings, hard scale or casting skin x 0.7-0.8
- Difficult to machine shapes x 0.8-0.9
- Easy to machine shapes x 1.05-1.20
- Boring x 0.75-0.85
- Interrupted cutting x 0.8-0.9
- Machine in good condition x 1.05-1.20
- Machine in poor condition x 0.8-0.95

H.S.S. CUTTING SPEEDS FOR DRILLING, TURNING & MILLING

Material to be cut	M/Min	Feet/Min
Alloy steels over 80 tons tensile	5-9	15-30
Mon magnetic stainless steels	6-12	20-40
Alloy steels 60-80 tons tensile	9-15	30-50
Magnetic stainless steels	12-18	40-60
Non alloy steels above 0.7% carbon	12-18	40-60
Alloy cast iron	12-21	45-70
Alloy steels up to 60 tons tensile	15-21	50-70
Non alloy steels 0.4-0.7% carbon	18-24	60-80
Grey cast iron	18-30	60-100
Non alloy steels up to 0.4% carbon	24-30	80-100
Free machining steels	30-45	100-150
Free machining brass & aluminium	30-61	100-250

To calculate rotational speed use the formula, 12 x cutting speed in ft/min, divided by 3 x the diameter (in inches) of the cutter or the work size in the case of lathe turning. Or 1000 x cutting speed in M/min, divided by 3 x the diameter (in MM) of the cutter or work size in the case of lathe turning.

REAMING SPEED & FEEDS

A general rule for machine reaming holes is to run at half the drilling speed and double the feed. It is important to drill the hole to a size that will not leave too much or too little material for removal by the reamer. A reamer can only follow the hole it is reaming. If your drilled hole is out of alignment your reamer will follow it!!

Dia. of Reamer	Max. allowance	Min. allowance
Up to 10mm	0.30mm	0.20mm
10-14mm	0.40mm	0.25mm
14-18mm	0.50mm	0.25mm
18-35mm	0.50mm	0.30mm
30-50mm	1.0mm	0.40mm
Up to 3/8”	0.010”	0.007”
3/8-3/4”	1/64”	0.010”
3/4-1.14”	0.020”	0.012”
1.1/4-2”	1/32”	1/64”