Machining

Recommended Tools and Machining Parameters Applicable to:
Corrosion-resistant Alloys
High-temperature Alloys

Be aware that the cobalt-base alloys in these categories (25 and 188) can require different feeds and speeds (as noted in the table) than the nickel- and iron-based alloys.

Operations	Tool Types	Tool Geometry and Set-Up	Speed	Feed	Depth of Cut	Lubricant
-	-	-	surface ft./min*	in**	in**	-
Roughing with severe interruptions; Turning or Facing	Carbide: C-2 or C-3 grade	Negative rake square or trigon insert, 45° SCEA 1, 1/16 in nose radius Tool holder: 5° negative back rake, 5° negative side rake	30-50	0.004-0.008 per revolution	0.15	Dry[2], oil[3], or water-based[4,5]
Normal roughing; Turning or Facing	As above	As above	90 (80 for cobalt alloys)[6]	0.010 per revolution	<0.15	Dry, oil, or water-based
Finishing; Turning or Facing	As above	Positive rake square or trigon insert, if possible, 45° SCEA[1], 1/32 in nose radius Tool holder: 5° positive back rake, 5° positive side rake	95-110 (90 for cobalt alloys)	0.005-0.007 per revolution	0.04	Dry or water-based
Rough Boring	As Above	If insert-type boring bar, use standard positive rake tools with largest possible SCEA and 1/16 in nose radius If brazed tool bar, grind 0° back rake, 10° positive side rake, 1/32 in nose radius and largest possible SCEA	70 (60 for cobalt alloys)	0.005-0.008 per revolution	0.125	Dry, oil, or water-based
		Use standard positive rake				

Finish Boring	As Above	tools on insert-type bars Grind brazed bars as for finish turning, except back rake may be best at 0°	95-110 (90 for cobalt alloys)	0.002-0.004 per revolution	0.04	Water-based
	High Speed Steel: M-2, M-7, or M-40 series7	Radial and axial rake 0° to 10° positive, 45° corner angle, 10° relief angle	20-30 (20-25 for cobalt alloys)	0.003-0.005 per tooth	-	Oil or water-based
	Carbide: C-2 grade (marginal performance)	Use positive axial and radial rake, 45° corner angle, 10° relief angle	50-60 (35-40 for cobalt alloys)	0.005-0.008 per tooth (0.005 per tooth for cobalt alloys)	-	Oil or water-based
End Milling	High Speed Steel: M-40 series or T-15	If possible, use short mills with four or more flutes for rigidity	20-25 (15-20 for cobalt alloys)	Feed per tooth: ¼ in dia. 0.002 ½ in dia. 0.002 ¾ in dia. 0.003 1 in dia. 0.004 (cobalt alloys: ¼ in dia. 0.001 ½ in dia. 0.0015 ¾ in dia. 0.002 1 in dia. 0.003)	-	Oil or water-based
	Carbide: C-2 grade	Use sharp tools with 4 or more flutes and variable lead, if possible	50-60 (40-50 for cobalt alloys)	As above	-	Oil or water-based
			10-15	Feed per rev.: â…› in dia. 0.001		

Drilling	High Speed Steel: M-33, M-40 series, or T-15	Use short, heavy-web drills with 135° crank shaft point; thinning of web at point may reduce thrust and aid chip control	(7-10 for cobalt alloys) Maximum of 200 rpm for ¼ in dia. drills or smaller	¼ in dia. 0.002 ½ in dia. 0.003 ¾ in dia. 0.005 1 in dia. 0.007 (same for cobalt alloys)	-	Oil or water-based Use coolant feed drills if possible
	Carbide: C-2 grade	Not recommended, but tipped drills may be successful on rigid set-ups if depth is not great. The web must be thinned to reduce thrust; use 135° included angle on point Gun drill can be used	50 (40 for cobalt alloys)	As above	-	Oil or water-based Coolant-fed, carbide-tipped drills may be economical in some set-ups
Reaming	High Speed Steel: M-33, M-40 series, or T-15	Use 45° corner angle, narrow primary land, and 10° relief angle	10-15 (8 for cobalt alloys)	Feed per rev.: ½ in dia. 0.003 2 in dia. 0.008 (same for cobalt alloys)	-	Oil or water-based
	Carbide: C-2 or C-3 grade	Tipped reamers recommended; solid reamers require very good set-up Tool geometry same as above	40 (20 for cobalt alloys)	As above	-	Oil or water-based
Tapping	High Speed Steel: M-1, M-7, or M-10	Use two flute, spiral point, plug tap 0° to 10° hook angles Nitrided surface may be helpful by increasing wear resistance, but may result in chipping or breakage Tap drill for 60-65% thread if possible, to increase tool	7 (same for cobalt alloys)	-	-	Use best possible tapping compound; sulfo-chlorinated oil-base preferred

		life				
	Carbide: not recommended	-	-	-	-	-
Electrical Discharge Machining	HAYNES® and HASTELLOY® alloys can be readily cut using any conventional Electrical discharge machining (EDM) system, or by wire EDM					

General note: Use high pressure coolant systems and through the tool coolant, when possible.

*To convert to surface m/min, multiply by 0.305
**To convert from in to mm, multiply by 25.4

[1]SCEA = side cutting edge angle, or lead angle of the tool
[2]At any point where dry cutting is recommended, an air jet directed at the tool may provide a substantial increase in tool life
[3]Oil coolants should be premium quality, sulfo-chlorinated oils, with extreme pressure additives; a viscosity of 50 to 125 SSU at 100°F (38°C) is standard
[4]Water-based coolants should consist of a 15:1 mixture of water and either a premium quality, sulfo-chlorinated, water-soluble oil or a chemical emulsion, with extreme pressure additives
[5]Water-based coolants may cause chipping or rapid failure of carbide tools in interrupted cuts
[6]Depending upon the rigidity of the set-up
[7]M-40 series high speed steels include M-41 through M-46 at time of writing; others may be added and should be equally suitable

Applicable to:
Wear & Corrosion-resistant Alloy

ULTIMET® alloy can be successfully turned, drilled, and milled if appropriate tooling and parameters are employed. However, the alloy possesses high strength and work hardens rapidly. Machining guidelines specific to ULTIMET® alloy are as follows:

Turning (ULTIMET® alloy)

Carbide (not high speed steel) tools are recommended.

Surface speed: 60-70 surface ft./min (0.30-0.35 m/s).

Feed rate: 0.005-0.010 in (0.13-0.25 mm).

Depth of cut for roughing: 0.05-0.10 in (1.3-2.5 mm).

Depth of cut for finishing: 0.010-0.015 in (0.25-0.38 mm).

Drilling (ULTIMET® alloy)

Carbide tipped or high speed steel drills are recommended.

Surface speed: 30-35 surface ft./min (0.15-0.18 m/s) for carbide tipped drills; 8-10 surface ft/min (0.04-0.05 m/s) for high speed steel drills.

Feed rate: 0.004 in (0.1 mm) per revolution for 0.25 in (6.4 mm) diameter and greater.

135° included angle on point.

Milling (ULTIMET® alloy)

Carbide (not high speed steel) end mills are recommended.

Surface speed: 25-30 surface ft/min (0.13-0.15 m/s).

Feed per tooth: 0.002 in (0.05 mm) for cutter diameters below 0.75 in (19 mm); 0.003 in (0.08 mm) for cutter diameters above 0.75 in (19 mm).