General Considerations

Cutting Compounds

Virtually any lubricant or coolant, or none at all, can be used in machining the alloys; in many cases, they respond well to ordinary sulfurized mineral oil. Sulfur imparts improved lubricity and anti-weld properties and is also believed to provide better chip action by embrittling the metal surface layer during chip formation when the tool edge-metal interface temperature becomes high enough.

In machining, if the temperature of the oil and workpiece becomes too high so that the sulfurized mineral causes brown sulfur staining, the stain can be readily removed with a proprietary cleaning solution of the sodium cyanide or chromic-sulfuric acid type. This should be done prior to any thermal treatment, including welding, because upon further exposure to elevated temperature the staining may cause intergranular surface attack. A second word of caution—the parts should not be immersed in cleaning solutions any longer than necessary to remove the stain. Prolonged exposure to some acid solutions will cause severe intergranular corrosion. In high-speed operations which are prone to create high temperatures, sulfur embrittlement might preclude the use of a sulfurized oil because of damage to cemented carbide tooling. (Many cemented carbides have a nickel or cobalt matrix which would be sensitive to sulfur attack at elevated temperatures.) However, flooding the cutting area with lubricant will generally cool the tool bit sufficiently to avoid breakdown of the carbide bond.

Water-base coolants are preferred for use in high-speed operations such as turning, milling and grinding because of their greater cooling effect. These may be soluble oils or proprietary chemical mixtures. Except for grinding, which depends almost entirely upon cooling and flushing, some chemical activity is desired, even in coolants, and is generally provided by chlorine, amines or other chemicals. Since chemical coolants are often proprietary compounds, assurance of chemical activity should be obtained from the manufacturer.

For slower operations such as drilling, boring, tapping and broaching, heavy lubricants and very rich mixtures of chemical coolants are desirable. Oils should be used when drilling Nickel 200 and INCONEL alloy X-750. In the drilling and tapping of small diameter holes and similar operations where lubricant flow and chip flushing are restricted, solvents will improve performance. These less viscous materials can be used alone or to dilute mineral and lard oils.

Spray mist coolant is adequate for simple turning operations on alloys of all groups.

Work Hardening

The nickel alloys have an austenitic matrix and, like the austenitic stainless steels, work harden rapidly. (See Figure 1.) The high pressures developed between the tool and workpiece during cutting or grinding produce a stressed layer of deformed metal on the surface of the work. The deformation causes a hardening effect that retards further machining. The stresses in this deformed layer not only affect the mechanical properties of the workpiece but also can cause distortion of parts that have small cross sections.

One method of reducing work hardening during machining is to work harden the material prior to machining, by cold working. Cold-drawn, stress-relieved material is always preferred for machining, particular when the smoothest finish is desired. Hot-rolled material is next best while annealed material is least desirable in most applications.

The best finish is produced on age-hardenable alloys by machining them in the aged condition. Because the high strength and hardness of aged material prevent heavy cuts, rough machining is done before age hardening. Solution annealing usually improves machinability of age-hardenable alloys by dissolving hard phases.

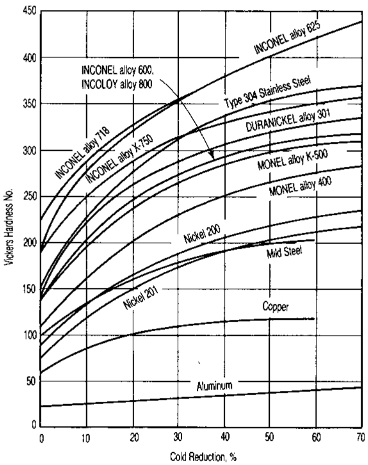


Figure 1. Degree of work hardening of some metals as indicated by the effect of cold reduction on hardness.

A second method of minimizing work hardening is employing careful machining practices. Sharp tools with positive rake angles, which cut metal instead of pushing it, are required. Feed rates and depths of cut must be sufficient to prevent burnishing or glazing. Tools should not be allowed to rub the work, either because of improper clearance or by being allowed to dwell in the cut.

Distortion and Stress Relieving

Even under the best of machining conditions, some stresses are produced that may cause subsequent distortion of the work. For maximum dimensional stability, it is best to rough out the material slightly oversize, stress relieve it, then finish it to size. Stress relieving has little effect on dimensions but may affect mechanical properties. Suitable temperatures and times for such heat treatment vary with the specific alloy and dimensions of the work. Guidelines for heat treatment are given in the technical bulletins which describe the alloys.

Microstructure

Grain size has little direct effect on machinability; indirectly it may have some effect since grain size does reflect thermal processing and change in constitution of structure. Some difference in surface finish may be noted but this can be minimized by correct cutting procedure.

In general, microstructure affects machinability in two ways:

1. The presence of graphite or sulfide phases will greatly enhance machinability.
2. Hard phases such as carbides, nitrides, carbonitrides, oxides, silicates and possibly also the gamma prime type phase Ni_3 (Al, Ti) are abrasive and tend to cause rapid tool wear.

The nickel-chromium and nickel-chromium-iron alloys are less abrasive than the common grades of austenitic stainless steel because they have lower carbon contents and therefore fewer carbides. Probably the hardest and most abrasive of all the phases is titanium carbide (TiC), which is present in most of the age-hardenable alloys. Another is niobium carbide (NbC). These, and certain other phases, are usually present in as-rolled or mill-annealed products. Solution annealing at high temperatures, 2000°F (1095°C) and above, is required to dissolve them. Age hardening tends to precipitate greater amounts of niobium and titanium carbides along with chromium carbides and the gamma prime phase. The refractory nature of these phases, together with their strengthening effect, makes age-hardened material less machinable although chip action is improved.

These phases have little or no measurable effect on finishes obtainable on the alloys. Softer and "gummier" alloys, such as those in the A, B, and C Groups identified below, will exhibit torn surfaces in roughing operations, and greater care must be exercised in finishing operations. With proper technique, however, finishes on the order of 4 to 8 microinches (0.1 to 0.2 μm) and finer can be obtained by grinding, honing and lapping. It may not always be possible to produce these finishes on MONEL alloy R-405 because of the presence of sulfide inclusions.

Classification of Alloys

Table 1 classifies alloys with respect to machining characteristics. Unless noted, all alloys in a given group require similar machining practices.

Group A consists of nickel alloys. These alloys are characterized by moderate mechanical strength and high degree of toughness. They can be hardened only by cold work. The alloys are quite gummy in the annealed or hot-worked condition, and cold-drawn material is recommended for best machinability and smoothest finish.

Group B consists mainly of those nickel-copper alloys that can be hardened only by cold work. The alloys in this group have higher strength and slightly lower toughness than those in Group A. Cold-drawn or cold-drawn, stress-relieved material is recommended for best machinability and smoothest finish.

Group C consists largely of the nickel-chromium and nickel-iron-chromium alloys. These alloys are quite similar to the austenitic stainless steels. They can be hardened only by cold work and are machined most readily in the cold-drawn or cold-drawn, stress-relieved condition.

Group D consists primarily of the age-hardenable alloys. It is divided into two subgroups:

D-1—Alloys in the unaged condition.

D-2—Aged Group D-1 alloys plus several other alloys in all conditions.

The alloys in Group D are characterized by high strength and hardness, particularly when aged. Material which has been solution annealed and quenched or rapidly air cooled is in the softest condition and does machine easily. Because of softness, the unaged condition is necessary for ease in drilling, tapping and all threading operations.

Heavy machining of the age-hardenable alloys is best accomplished when they are in one of the following conditions:

1. Solution annealed.
2. Hot worked and quenched or rapidly air cooled.

The publications describing the various alloys should be consulted for details on the thermal processes involved. Although fully age-hardened material is usually too hard for tools with weak cutting edges, such as small drills and taps, and also for rough machining, material in this condition can be finish-machined to fine finishes and close tolerances.

The best way to machine the alloys of Group D, therefore, is to machine slightly oversize in the unaged condition, age-harden, then finish to size. Because the age-hardening treatment will relieve machining stresses, allowance must be made for possible warpage. A slight permanent contraction (up to about 0.07%) takes place during aging. Aged material has good dimensional stability.

Accurate dimensions and a good finish will result from following these practices. Work which is aged after finishing to size may not meet tolerances because of dimensional change upon the relief of machining stresses and also because of the formation of oxide. The scale would require pickling or polishing.

Group E contains one member, MONEL alloy R-405. It is designed for high production rates on automatic screw machines and is discussed separately in the section Automatic Machining.

Table 1 —Classification of Alloys by Machining Characteristics

Alloy	Nominal Chemical Composition,[a] %
GROUP A	
Nickel 200	Ni 99.6, C 0.08
Nickel 201	Ni 99.6, C 0.01
Nickel 205	Ni 99.6, C 0.04, Mg 0.04
Nickel 212	Ni 97.0, C 0.05, Mn 2.0
Nickel 222	Ni 99.5, Mg 0.075
GROUP B	
MONEL alloy 400	Ni 66.5, Cu 31.5
MONEL alloy 401	Ni 42.5, Cu 55.5, Mn 1.6, Fe 0.3
MONEL alloy 450	Ni 30.0, Cu 68.0, Fe 0.7, Mn 0.7
FERRY alloy	Ni 45.0, Cu 55.0
NILO alloy 36	Ni 36.0, Fe 64.0
NILO alloy 48	Ni 48.0, Fe 52.0
NILO alloy K	Ni 29.5, Fe 53.0, Co 17.0
UDIMAR alloy 250	Ni 19.0, Fe 76.0, Mo 3.0, Ti 1.4
GROUP C	
Nickel 270	Ni 99.98, C 0.01
MONEL alloy K-500, Unaged	Ni 65.5, Cu 29.5, Al 2.7, Fe 1.0, Ti 0.6
INCONEL alloy 600	Ni 76.0, Cr 15.5, Fe 8.0
INCONEL alloy 601	Ni 60.5, Cr 23.0, Fe 14.0, Al 1.4
INCONEL alloy 690	Ni 61.0, Cr 29.0, Fe 9.0
NIMONIC alloy 75	Ni 80.0, Cr 19.5
NIMONIC alloy 86	Ni 64.0, Cr 25.0, Mo 10.0, Ce 0.03
INCOLOY alloy 800	Ni 32.5, Fe 46.0, Cr 21.0, C 0.05
INCOLOY alloy 800HT	Ni 32.5, Fe 46.0, Cr 21.0, C 0.08, Al+Ti 1.0
INCOLOY alloy 802	Ni 32.5, Fe 46.0, Cr 21.0, C 0.4
INCOLOY alloy 825	Ni 42.0, Fe 30.0, Cr 21.5, Mo 3.0, Cu 2.2, Ti 1.0
INCOLOY alloy DS	Ni 37.0, Fe 41.0, Cr 18.0, Si 2.3
INCOLOY alloy 330	Ni 35.5, Fe 44.0, Cr 18.5, Si 1.1
INCOLOY alloy 020	Ni 35.0, Fe 37.0, Cr 20.0, Cu 3.5, Mo 2.5, Nb 0.6

Alloy	Nominal Chemical Composition,[a] %
GROUP D-1	
DURANICKEL alloy 301, Unaged	Ni 94.0, Al 4.4, Ti 0.6
INCOLOY alloy 925	Ni 42.0, Fe 32.0, Cr 21.0, Mo 3.0, Ti 2.1, Cu 2.2, Al 0.3
INCOLOY alloy MA 956	Fe 74.0, Cr 20.0, Al 4.5, Ti 0.5, Y_2O_3 0.5
NI-SPAN-C alloy 902, Unaged	Ni 42.5, Fe 49.0, Cr 5.3, Ti 2.4, Al 0.5
GROUP D-2	
DURANICKEL alloy 301, Aged	Ni 94.0, Al 4.4, Ti 0.6
MONEL alloy K-500, Aged	Ni 65.5, Cu 29.5, Al 2.7, Fe 1.0, Ti 0.6
INCONEL alloy 617	Ni 52.0, Cr 22.0, Mo 9.0, Co 12.5, Fe 1.5, Al 1.2
INCONEL alloy 625	Ni 61.0, Cr 21.5, Mo 9.0, Nb 3.6, Fe 2.5
INCONEL alloy 706	Ni 42.0, Fe 36.5, Cr 16.0, Ti 1.8, Nb 3.1
INCONEL alloy 718	Ni 54.0, Cr 18.0, Fe 18.5, Nb 5.1, Mo 3.0
INCONEL alloy X-750	Ni 73.0, Cr 15.5, Fe 7.0, Ti 2.5, Nb 1.0, Al 0.7
INCONEL alloy 751	Ni 73.0, Cr 15.5, Fe 7.0, Ti 2.5, Al 1.1, Nb 1.0
INCONEL alloy MA 754	Ni 77.5, Cr 20.0, Fe 1.0, Ti 0.5, Al 0.3, Y_2O_3 0.6
NIMONIC alloy 80A	Ni 76.0, Cr 19.5, Ti 2.4, Al 1.4
NIMONIC alloy 81	Ni 67.0, Cr 30.0, Ti 1.8, Al 0.9
NIMONIC alloy 90	Ni 60.0, Cr 19.0, Co 16.5, Ti 2.5, Al 1.5
NIMONIC alloy 105	Ni 54.0, Cr 15.0, Co 20.0, Mo 5.0, Al 4.7, Ti 1.3
NIMONIC alloy 115	Ni 60.0, Cr 14.2, Co 13.2, Al 4.9, Ti 3.8, Mo 3.2
NIMONIC alloy 263	Ni 51.0, Cr 20.0, Co 20.0, Mo 5.8, Ti 2.2, Al 0.5
NIMONIC alloy 901	Ni 42.5, Fe 36.0, Cr 12.5, Mo 5.8, Ti 2.9
NIMONIC alloy PE11	Ni 39.0, Fe 34.0, Cr 18.0, Mo 5.2, Ti 2.3, Al 0.8
NIMONIC alloy PE16	Ni 43.5, Fe 34.0, Cr 16.5, Mo 3.3, Ti 1.2, Al 1.2
NIMONIC alloy PK50	Ni 58.0, Cr 19.5, Co 13.5, Mo 4.25, Ti 3.0, Al 1.4
INCOLOY alloy 903	Ni 38.0, Fe 41.5, Co 15.0, Nb 3.0, Ti 1.4, Al 0.9
INCOLOY alloy 907	Ni 38.4, Fe 42.0, Co 13.0, Nb 4.7, Ti 1.5, Al 0.03, Si 0.15
INCOLOY alloy 909	Ni 38.4, Fe 42.0, Co 13.0, Nb 4.7, Ti 1.5, Al 0.03, Si 0.4
NI-SPAN-C alloy 902, Aged	Ni 42.5, Fe 49.0, Cr 5.3, Ti 2.4, Al 0.5
INCOLOY alloy G-3	Ni 44.0, Fe 19.5, Cr 22.0, Mo 7.0, Cu 2.0, Co 2.5
INCOLOY alloy C-276	Ni 57.0, Cr 15.5, Mo 16.0, Fe 5.5, W 3.8, Co 1.2, Mn 0.5
INCOLOY alloy HX	Ni 47.5, Fe 18.5, Cr 21.8, Mo 9.0, Co 1.5
GROUP E	
MONEL alloy R-405	Ni 66.5, Cu 31.5, Fe 1.2, Mn 1.1, S 0.04

[a] Not for specifications.

Turning

Tools

Cemented carbide tools produce the highest cutting rates and are recommended for most turning operations involving uninterrupted cuts.

Cast alloy tools are recommended for turning Group A alloys at optimum cutting rates. As with cemented carbide tools, interrupted cutting is not included in this recommendation.

High-speed-steel tools should be used for interrupted cuts such as occur in roughing of uneven surfaces. They also are used for finishing to close tolerances, finishing with smoothest surfaces and cutting with the least amount of work hardening.

Tables 2 and 3 respectively list various grades of cemented carbide and high-speed steels.

The performance of high-speed steel and cast alloy tools in cutting INCONEL alloy X-750 was evaluated in the laboratory. The material was in the hot-rolled condition. The tests were made in the lab turret lathe at 0.100-inch (2.54-mm) depth of cut and 0.01175 ipr (0.2985 mm/rev.) feed rate. Figure 2 shows that the cutting speeds used with the cast alloys are higher than those used with the high-speed steels. For equivalent cutting time, the cast alloy tools must be operated at higher cutting speeds.

The cast alloys tested had much higher resistance to abrasion and heat than the HSS tools, but chipping was excessive and very low cutting speeds were required to maintain adequate tool life. The high-speed steel tools exhibited low resistance to abrasion and heat but high resistance to chipping. The 18-4-1 tungsten HSS tool had lower resistance to chipping and abrasion and heat than the molybdenum-cobalt types.

Tool Geometry

Figure 3 illustrates the standard nomenclature for single-point turning tools and the recommended geometry for use with the various alloys. It is important that the tools have positive rake angles. This ensures that the metal is "cut" instead of "pushed," as would occur if negative rake angles were used. A secondary function of the rake angles is to guide the chip away from the finished surface. The side and end relief angles provide clearance between the tool flank and the work. The relief angles chosen (Figure 3) are a compromise between two factors. Too small an angle between the end of the tool and the work will cause the tool to rub against the workpiece as soon as a little tool wear has occurred. This will cause surface work hardening. Too large an angle will weaken the support of the cutting edge.

The same principles apply to the end cutting

Table 3—High-Speed Steels[a]

Type	Symbol	Nominal Chemical Composition, % Tungsten	Molybdenum	Chromium	Vanadium	Cobalt
Molybdenum	M-1	1.5	8.0	4.0	1.0	—
	M-2	6.0	5.0	4.0	2.0	—
	M-3	6.0	5.0	4.0	2.7	—
	M-4	5.5	4.5	4.0	4.0	—
	M-6	4.0	5.0	4.0	1.5	12.0
	M-7	1.75	8.75	4.0	2.0	—
	M-8	5.0	5.0	4.0	1.5	—[b]
	M-10	—	8.0	4.0	2.0	—
	M-15	6.5	3.5	4.00	5.0	5.0
	M-30	2.0	8.0	4.0	1.25	5.0
	M-34	2.0	8.0	4.0	2.0	8.0
	M-35	6.0	5.0	4.0	2.0	5.0
	M-36	6.0	5.0	4.0	2.0	8.0
Tungsten	T-1	18.0	—	4.0	1.0	—
	T-2	18.0	—	4.0	2.0	—
	T-3	18.0	—	4.0	3.0	—
	T-4	18.0	—	4.0	1.0	5.0
	T-5	18.0	—	4.0	2.0	8.0
	T-6	20.0	—	4.5	1.5	12.0
	T-7	14.0	—	4.0	2.0	—
	T-8	14.0	—	4.0	2.0	5.0
	T-9	18.0	—	4.0	4.0	—
	T-15	12.0	—	4.00	5.0	5.0

[a] American Iron and Steel Institute—(1955).

[b] 1.25% niobium.

Table 2—Cemented Carbides[a]

MANUFACTURER

CISC[b] GRADE	Adamas	Carboloy	Carmet	Firlomet	Firthite	Kenna-Metal	Newcomer	Sandvik Coromant	Talide	Tungsten Alloy	Valenite	Vascoloy Ramet	Wesson	Willey
C-1	8	44A	CA3	FA5	H	K1	NC4	H1	C89	9	VC1	2A68, VR54	GS	E8, E13
C-2	A	883, 860	CA4	FA6	HA	K6	NC3	H1	C91	9H	VC2	2A5, VR54	G1	E6
C-3	AA	905	CA7	FA7	HE	K8	NC2	H3	C93	9C	VC3	2A7	GA	E5
C-4	AAA	999	CA8	FA8	HF	K11	NC2	H5	C95	9B	VC4	2A7	GF	E3
C-5	DD	78C	CA51	FT3	TQ4	KM	NS65, NS4	S6, S4	S88	11T	VC5	EE, VR77	WS	945
C-5A	434	370	CA610	FT41, FT5	TXH	K21	—	S1P	S88X	9S	VC125	VR77, VR75	26	8A
C-6	D	78B	CA609	FT4	TXH, TA	K2S	NS3	S2	S90	10T	VC6	VR75	WM	710
C-7	C	78	CA608	FT6	TXL	K5H	NS2, NS17	S1	S92	8T	VC7	E, VR73	WH	606
C-7A	548	350	CA606	FT61	T16, TXL	K4H	—	—	S92X	5S	—	VR73	WH	6A
C-8	CC	330	CA605	FT7	T31, WF	K7H	NS15	F1	S94	5S	VC8	EH	WH	509, 4A

[a] From DMIC Memorandum 134 (1961).

[b] Carbide Industry Standardization Committee.

For the same CISC grade, there seem to be no truly equivalent carbides of different brands. Where two carbide grades from the same manufacturer are shown for the same CISC grade, the first is sometimes recommended.

This chart can function only as a guide. The so-called "best grade" may differ for each specific job even if the material being machined is the same. The final selection can be made only by trial and error. Instructions regarding the specific use and application of any competitive grade should be obtained directly from the manufacturer.

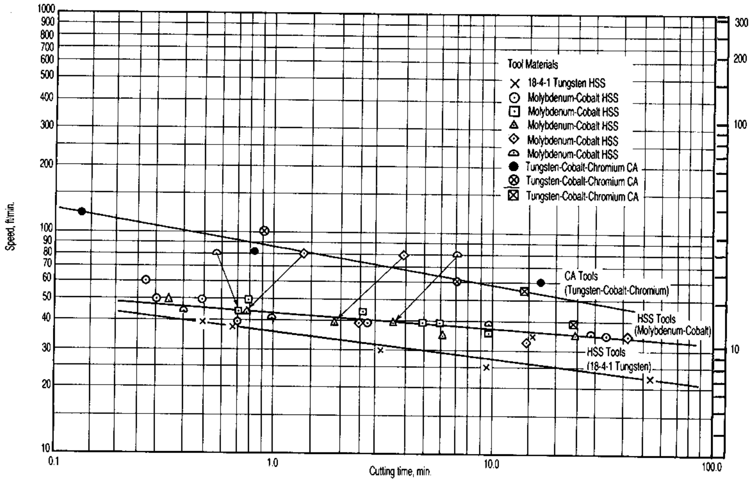
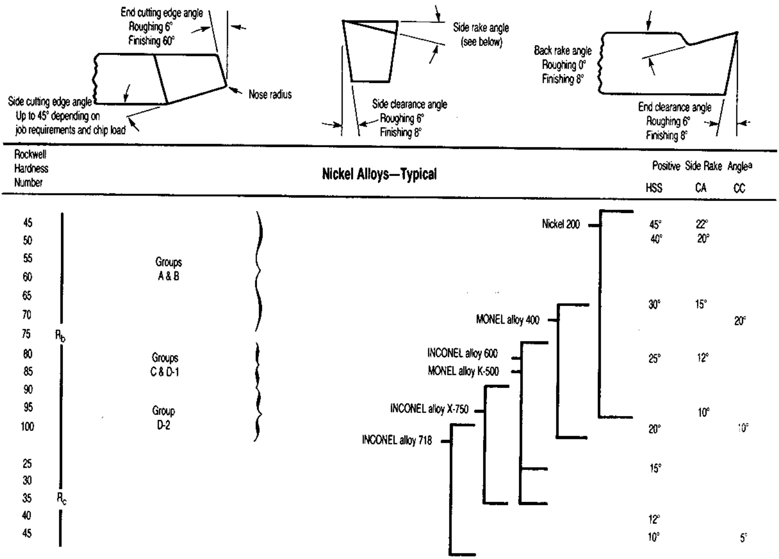


Figure 2. Cutting speeds and times for turning INCONEL alloy X-750 with various tool materials.



[a]Finishing cuts and light to medium-heavy roughing cuts Feed rates—up to 0.025 ipr. (0.006 mm/rev.) Depth of cuts—up to 0.250 in. (6.35 mm)

Figure 3. Recommended geometry for single-point turning tools.

edge angle. This angle supports the nose of the cutting tool by resisting the forces of tool feed.

The side cutting edge angle is second in importance only to the rake angle. The thickness of the chip is controlled to some extent by the size of this angle because it affects distribution of the load on the cutting edge. It also, in conjunction with the rake angle, provides directional control to the chips.

The nose radius, which joins the end and side cutting edges, provides strength to the tool nose and helps dissipate the heat generated by the cut. The scalloped effect produced by a tool with a nose radius (Figure 4) gives a better surface finish, shallower scratches and a stronger workpiece with less tendency to crack at sharp corners than the notched effect produced by a sharp tool. Figure 5 is a guide for sizing nose radii. Oversized radii can interfere with the cutting action, causing tool vibrations which tend to work harden the machined surface and reduce tool life.

Cutting Compounds

Coolants, either the chemical type or the oil emulsion type, should be used for all roughing operations and finish cuts with carbide tools. Any work done with high-speed-steel tools can be improved by the use of sulfurized, chlorinated cutting oil.

Cutting Feeds and Speeds

Ranges of cutting speeds and feeds appear in Table 4. The centers of the ranges should be used as starting points in establishing the best conditions for specific jobs.

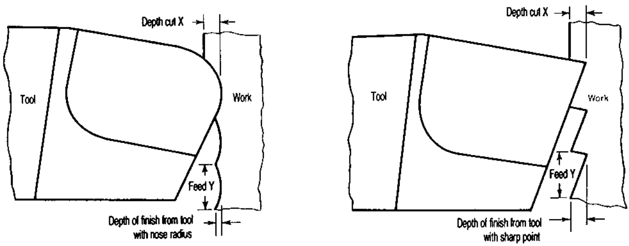


Figure 4. Effect on work by tool with nose radius.

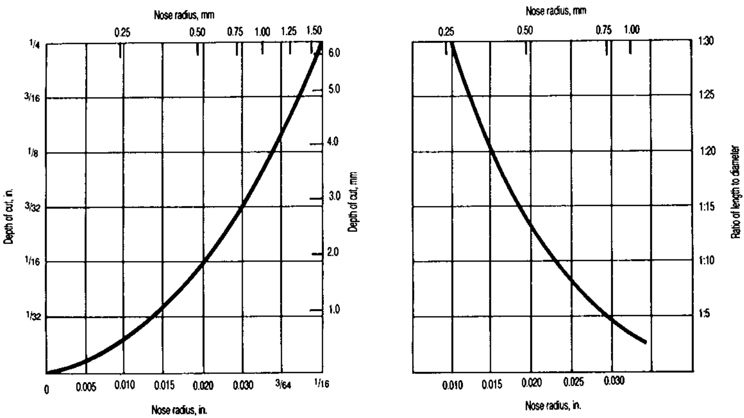


Figure 5. Guide for sizing nose radius. Use right-hand chart for machining long, slender work.

Chip Control

The alloys will present a minimum of chip disposal problems when cut with tools equipped with properly designed chip curlers or breakers. High-speed-steel tools require chip curlers, commonly referred to as lipped tools. The lip should include the proper rake angles for the alloy and should be wide and deep enough to cause the chip to curl and break but not to force it into a wad or tight knot.

Carbide tools should be ground or provided with chip breakers. With the latter, tool rake angles are plane surfaces that terminate at the chip breaker wall. The radius joining the chip breaker wall and the rake angle plane must be kept very small. It is important that the angle between the two surfaces be 125° to 135°. A small radius and the proper angle will tend to prevent the chip from welding in the chip breaker.

Width and depth of the chip breaker will vary depending on the feed rate used (see Table 5 and Figure 6).

Table 4—Conditions for Turning with Single-Point Tools

Group (any condition)	Hardness	Depth of Cut		Tool Material: High Speed Steel						Tool Material: Cast Alloy					Tool Material: Carbide							
				Surface Speed		Feed		Tool Material	Cutting Fluid[b]	Surface Speed		Feed		Cutting Fluid[b]	Surface Speed: Brazed Tool		Surface Speed: Throw Away		Feed		Tool Material	Cutting Fluid[b]
		in.	mm	fpm	m/min.	ipr	mmpr			fpm	m/min.	ipr	mm/rev.		fpm	m/min.	fpm	m/min.	ipr	mmpr		
A	45 R_b	0.250	6.35	50/60	15/18	0.030	0.762	T-5	II	a	a	a	a	a	a	a	a	a	a	a	a	a
	95 R_b	0.050	1.27	170/200	52/61	0.008	0.203	M-36	II, III	370/400	115/120	0.008	0.203	II, III	a	a	a	a	a	a	a	a
B	65 R_b	0.250	6.35	60/70	18/21	0.030	0.762	T-5	II	a	a	a	a	a	250/300	75/90	275/325	85/100	0.020	0.508	C-6	II
	100 R_b	0.050	1.27	90/100	27/30	0.010	0.254	M-36	II	150/200	45/60	0.008	0.203	II	300/350	90/110	350/400	110/120	0.008	0.203	C-7	II
C	75 R_b	0.250	6.35	25/35	7.6/11	0.030	0.762	T-5	II	a	a	a	a	a	150/200	45/60	175/225	55/70	0.020	0.508	C-6	II
	30 R_c	0.050	1.27	50/60	15/18	0.010	0.254	M-36	II	100/150	30/45	0.008	0.203	a	325/375	100/115	375/425	115/225	0.008	0.203	C-2	II
D-1	80 R_b	0.250	6.35	40/50	12/15	0.030	0.762	T-5	II	a	a	a	a	a	175/225	55/70	200/250	60/75	0.020	0.508	C-6	II
	35 R_c	0.050	1.27	60/70	18/24	0.010	0.254	M-36	II	100/150	30/45	0.008	0.203	II	225/275	70/85	250/300	75/90	0.008	0.203	C-8	II
D-2	85 R_b	0.250	6.35	12/18	3.7/5.5	0.010	0.254	T-5	II	a	a	a	a	a	30/40	10/12	40/60	12/18	0.010	0.254	C-2	II
	45 R_c	0.050	1.27	15/20	4.6/6.1	0.008	0.203	M-36	II	a	a	a	a	a	40/50	12/15	50/100	15/30	0.008	0.203	C-2	II
E	65 R_b	0.250	6.35	70/80	21/24	0.030	0.762	T-5	II	a	a	a	a	a	250/300	75/90	275/325	85/100	0.020	0.508	C-6	II
	100 R_b	0.050	1.27	120/130	37/40	0.010	0.254	M-36	II	175/225	55/70	0.008	0.203	II	300/350	90/110	350/400	110/120	0.008	0.203	C-7	II

[a] Not Recommended.

[b] II—Water base—oil emulsion or chemical solution;
III—Sulfurized or chlorinated oil or mixed oils.

Table 5—Typical Dimensions for Chip Breakers[a]

Feed Rate		Depth		Width	
ipm	mm/min.	in.	mm	in.	mm
0.005	0.127	0.015	0.381	0.060	1.52
0.010	0.254	0.020	0.508	0.080	2.03
0.020	0.508	0.030	0.762	0.150	3.81

[a] May vary slightly with alloy.

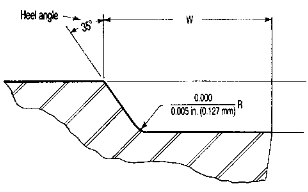


Figure 6. Chip breaker.

Drilling

Drilling is a complex operation, involving extrusion of metal by the chisel edge in the center of the drill and shear cutting by the lips of the tool. Because of high strength and work-hardening tendencies, drilling of high nickel alloys can be a difficult operation if good drilling practice is not used.

It is important that steady feed rates be employed. If the drill is allowed to dwell, it will cause excessive work hardening of the metal at the bottom of the hole. This will make it more difficult to resume cutting and may result in breaking of the drill when it does take hold.

The setup should be as rigid as possible. Stub drills are recommended. Drilling jigs with guide bushings should be used whenever possible.

Conventional high-speed drills are satisfactory for general purpose drilling of the alloys of Groups A and B. This type of drill, illustrated in Figure 7, has a point angle of 118°, a helix of about 30°, a 12° lip relief angle and a chisel edge angle of 125–135°.

Heavy-duty, high-speed drills with a heavy web are recommended for drilling the alloys of Groups C and D. Cobalt high-speed-steel drills give longer tool life. Cutting pressures will be reduced and a

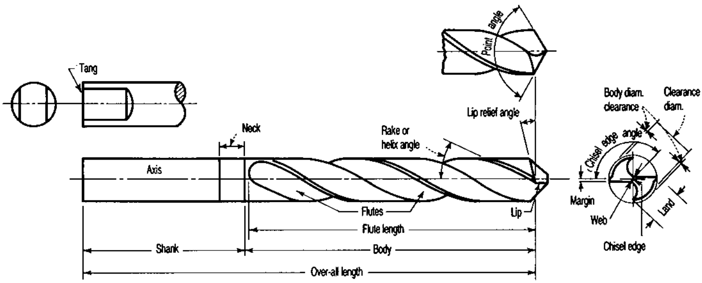


Figure 7. Conventional high-speed drill.

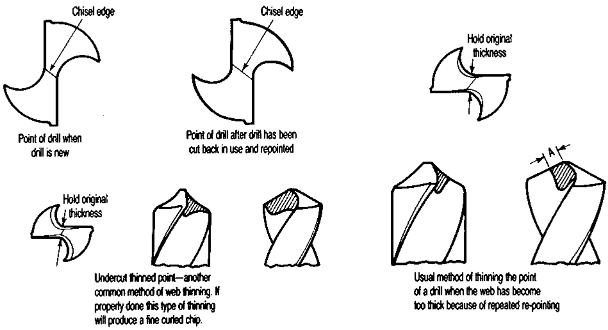


Figure 8. Thinning drill point.

positive effective rake will be maintained if the web is thinned at the chisel point as in Figure 8.

Crankshaft drills are useful for producing deep holes. These drills have a heavy web and a helix angle slightly higher than normal; the web is thinned at the chisel point, as shown in Figure 9.

Cutting action with drills larger than 3/4 inch (19 mm) in diameter will be improved by grinding several small grooves through the lip extending back along the lip clearance. The spacing of the grooves should be staggered between the two cutting edges (Figure 9). The effect of this serration will be to produce narrow chips with less tendency to foul in the helical flutes.

Suggested feeds and speeds for drilling with twist drills are given in Table 6.

Spade drills, such as that shown in Figure 10, are regularly used for deep-hole and heavy drilling, 1 1/2 inches (40 mm) in diameter and greater. The drill is secured in a steel head (Figure 11) which is attached to a rigid bar with bearing support between the work and tail stock. Spade drills are made of high-speed steel. Their cutting edges may be tipped with cemented carbide. Lead holes should

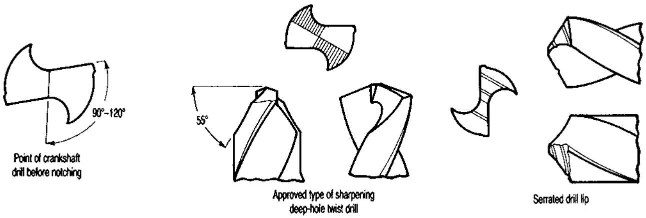


Figure 9. Variations in drills.

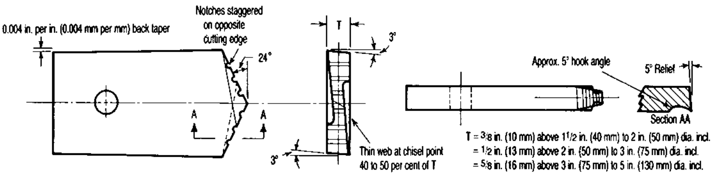


Figure 10. Spade drill.

Table 6—Drilling Conditions

Drill Diameter		Feed[a]		Alloy Group	Surface Speed	
in.	mm	ipr	mm/rev.		fpm	m/min.
Under 1/16	Under 1.6	0.0005–0.001	0.0127–0.025	A	55–75	17–23
1/16–1/8	1.6–3.2	0.001–0.002	0.025–0.051	B	45–55	14–17
1/8–3/16	3.2–4.8	0.002–0.004	0.051–0.102	C	25–35	8–11
1/4–5/16	6.4–7.9	0.003–0.005	0.076–0.127	D-1	20–30	6–9
3/8–7/16	9.5–11	0.004–0.007	0.102–0.178	D-2, Unaged	10–12	3–4
1/2–11/16	13–17	0.006–0.010	0.152–0.254	D-2, Aged	8–10	2–3
3/4–1	19–25	0.008–0.015	0.203–0.381	E	50–70	15–21

[a] Use the lower value for smaller drills in the range or for harder material.

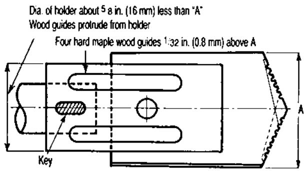


Figure 11. Spade-drill holder.

be made with a drill whose point is smaller than that of the spade drill. Table 7 shows feeds and speeds used for spade drilling.

Gun drills are also used for deep-hole drilling. Figure 12 shows three types of drills available. The manufacturer of the equipment should be consulted for correct practice on high nickel alloys.

Gun drills are used mostly for producing deep holes of diameters up to and including 2 inches (50 mm) but are occasionally used for holes up to 2 1/2 inches (60 mm) in diameter. For gun drills with insert tips, the speeds cited in Table 8 are appropriate. For high-speed-steel tools, the speeds shown in Table 6 should be used.

Feeds of 0.0005 to 0.003 ipr (0.0127 to 0.076 mm/rev.) are recommended for both high-speed-steel and carbide-tipped drills, using the lower feed for the smaller diameter holes. The proper feed for the job must be determined from the chip-breaking characteristics.

Table 7—Spade Drilling, 1–2 in. (25–51 mm) diam; tool material, M-2

Alloy Group	Surface Speed		Feed	
	fpm	m/min.	in.	mm
A	55–75	17 –23	0.005–0.007	0.127–0.178
B	45–55	14 –17	0.005–0.007	0.127–0.178
C	25–35	7.6–11	0.005–0.007	0.127–0.178

Table 9 lists insert materials which have been used successfully for tips on gun drills.

A highly sulfurized oil should be used for deep hole and all gun drilling operations. Lubricant pressures should be about 800 psi (6 MPa) for 3/16-inch (5-mm) holes, decreasing to around 200 psi (1.5 MPa) for 2-inch (50-mm) holes.

Table 8—Feeds and Speeds for Gun Drills[a]

Alloy Group	Feed		Surface Speed	
	ipr	mm/rev.	fpm	m/min.
A	0.0001 –0.002	0.0025 –0.051	220	67
B	0.00015–0.004	0.00381–0.102	300	91
C	0.0002 –0.005	0.0051 –0.127	320	98
D-1	0.0001 –0.003	0.0025 –0.076	220	67
D-2—Unaged	0.0001 –0.003	0.0025 –0.076	100	30
D-2—Aged	0.0001 –0.003	0.0025 –0.076	60	18

a For drill sizes 1/16–2 in. (1.6–50 mm).

Table 9—Insert Tips for Gun Drills

Alloy Group	Insert Material
A	Cast Alloy (30 Cr—18.5 W—40 Co)
B	Cemented Carbide—Type C-6
C	Cemented Carbide—Type C-2
D-1	Cemented Carbide—Type C-2
D-2	Cemented Carbide—Type C-2

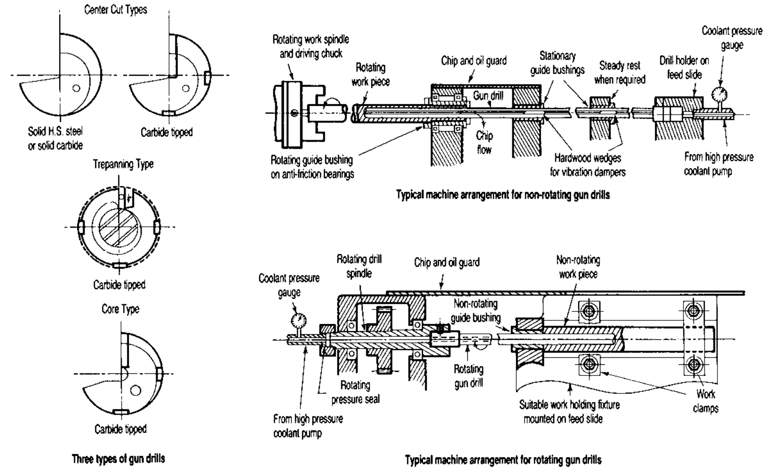


Figure 12. Gun drills and drilling set-ups.

Broaching

A well designed broach for a specific application and alloy must reflect a definite relationship between the pitch selected for the teeth, the feed (or step per tooth) and the cutting angles of the teeth. High-speed steels (types T-1, T-4 and M-4) are suitable materials for broaches. They should be hardened and tempered to about 64 Rockwell C.

Broaches should be kept sharp and reground at the first sign of dulling. The teeth should be polished or honed to remove all peaks and irregularities left by the grinding wheel.

Figure 13 illustrates a typical broach. Chip breakers, commonly referred to as nicked teeth, are recommended. They should be staggered from tooth to tooth to avoid overlapping. The nicks should be slightly larger than the depth of cut.

Although Group D alloys are broached more cleanly in the age-hardened condition, high forces are required for these materials in either the aged or unaged conditions. For example, when broaching the fir tree profiles in a jet engine turbine wheel of solution-annealed INCONEL alloy X-750 (hardness 93 Rockwell B), forces of 8–6 tons (71–53 kN) were required for the roughing teeth of the broach and 4–3 tons (36–27 kN) for the finishing teeth. When broaching the same section in fully age-hardened INCONEL alloy X-750, (hardness 31 Rockwell C), forces of 6–3 tons (53–27 kN) were required. The decrease in power requirements for the harder material was attributed to cleaner cutting and less drag on the broach.

A generous supply of cutting lubricant should be supplied to the broach and work, with every precaution being taken to ensure that the fluid flows to the cutting edges of the teeth. The oil should be free-flowing and of sufficient body to provide good lubricity to the chips. Sulfurized mineral oil is recommended.

Broaching speeds and rake angles appear in Table 10.

Table 10—Broaching Speeds and Rake Angles

Alloy Group	Surface Speed fpm	Surface Speed m/min.	Rake Angle, degrees
A	10–18	3.1–5.5	12–18
B	10–18	3.1–5.5	12–18
C	5–12	1.5–3.7	10–15
D-1	5–12	1.5–3.7	10–15
D-2	6	1.8	8–10

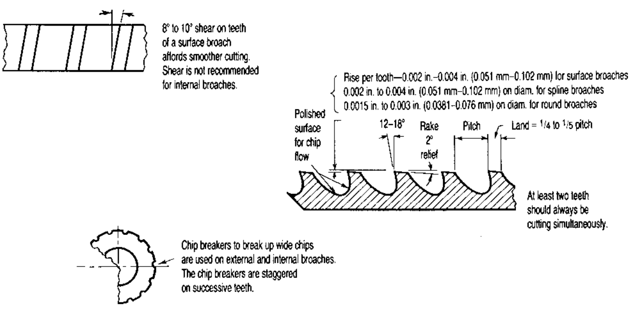


Figure 13. Broach.

Threading

Lathe Threading

Thread-cutting tools are ground according to the principles described for turning tools. A typical tool geometry is shown in Figure 14. The side rake of the tool is designed to provide easy cutting and a good finish. The angles on threading tools are smaller than on turning tools so that the small nose of the threading tool is well supported. The weakness of the tool nose and the small volume of tool available to dissipate heat make it necessary to machine at lower speeds and feeds than those used for straight turning. Standard single-point lathe threading practices are adequate for threading the nickel alloys.

The tool nose should be flooded with sulfurized oil during threading. If the machine is not equipped to pump oil on the work, the workpiece should be brushed with a sulfur-base oil during the cutting operation.

Threading speeds are listed in Table 11. The depth of cut will vary, becoming less as the work progresses and more of the tool cutting edge is engaged in removing metal.

Table 11—Threading Speeds, Single Point or Die Head

Alloy Group	Surface Speed	
	fpm	m/min.
A, B, E	25 -30	7.6 -9.1
C, D-1	12 -18	3.7 -5.5
D-2	3.0- 3.5	0.91-1.06

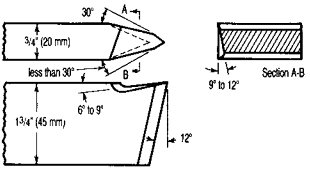
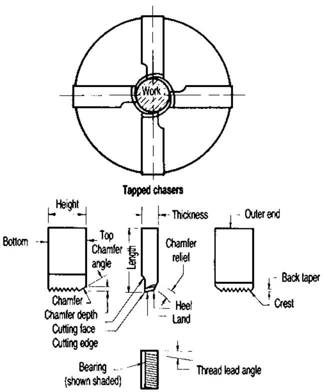


Figure 14. Threading Tool.

Die Head Threading

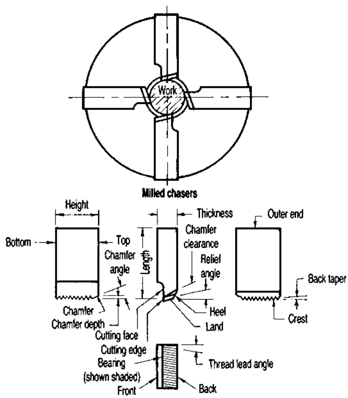
The most economical method of producing quantities of male threads is with self-opening dies. Threads may be cut with solid dies but they may be torn while reversing the die.

Threading dies should be made of molybdenum high-speed steel (Grade M-2 or M-10). Typical dies are shown in Figures 15 through 18. Dies must be kept sharp and flooded with lubricant (sulfurized



Die head chasers, blade type, tapped

Figure 15. Tapped die head chasers.



Die head chasers, blade type, milled

Figure 16. Milled die head chasers.

oil or rich mixture of soluble oil or chemical type coolant) during use.

A chaser throat (chamfer) angle of 15° to 20° is recommended for producing V threads where no shoulder is involved. When close-to-shoulder threading must be done, the die manufacturer's recommendations for the proper angle should be followed. The rake angle is ground 15° for alloys of Groups D-1 and D-2 and 30° for alloys of Groups A, B, C, and E.

The work diameter should be reduced by 1% to 1 1/2% to prevent binding in the die. The high ductility of the alloys permits metal to flow into the grooves of the die so that scanty threads will not be produced unless clearance is too great. The exact undersize required will vary with various alloys and tempers but can be determined by one or two trials.

Speeds shown in Table 11 for lathe threading apply also to die threading.

Thread Grinding

External threads may be produced on Group D-2 alloys (any condition) by form grinding. Aluminum oxide (150–320 grit) vitrified-bonded grinding wheels (medium hard, open structure) are used. The recommended coolant for thread grinding is a high-grade grinding oil of about 300 seconds viscosity at 70°F (21°C); it should be filtered. During grinding, extreme care must be taken to prevent overheating and resulting heat checking on the ground threads.

Thread Rolling

Threads can be rolled readily on material having hardness up to about R_c 30 and on a limited basis up to about R_c 40. Austenitic alloys work harden; Figure 1 indicates the degree of expected die life and rolling pressures that may be expected, as relative to various metals.

Maximum tensile properties may be obtained in the age hardenable (D-Group) alloys by thread rolling after direct aging. Usually it is preferable, however, to thread roll as-drawn or annealed material and then age harden. Material in the soft condition is more easily threaded and subsequent aging tends to stress relieve the cold-worked threads.

By using centerless ground blanks, threads can be obtained with dimensional tolerances superior to cut threads. Tangent, flat or cylindrical methods of thread rolling are acceptable. Detailed procedures should be obtained from the manufacturer of the thread rolling die.

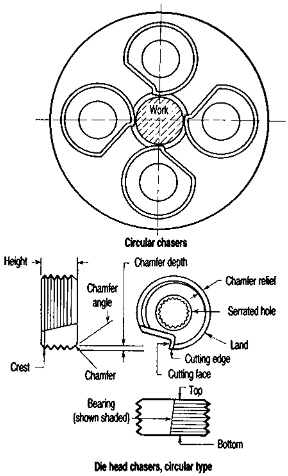


Figure 17. Circular die head chasers.

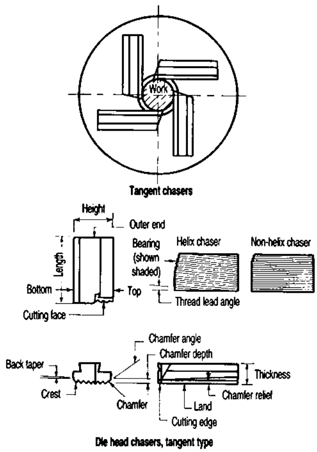


Figure 18. Typical tangential thread chasers.

14 Reaming

Standard fluted reamer nomenclature is shown in Figure 19. Fluted reamers for high nickel alloys are produced as standard items and can be identified as follows:

1. High-speed-steel tool material
2. Right-hand cut
3. Right-hand helix (positive axial rake)
4. Positive radial rake

Operating speeds for reamers should be 2/3 of the speed used for drilling the same material (see Table 6) but not so high as to cause chatter. Other factors contributing to chatter are lack of rigidity in set-up, misalignment and dull tools.

The reamer feed into the work should be 0.0015 to 0.004 inch (0.0381 to 0.102 mm) per flute per revolution. Too low a feed rate will result in glazing and excessive wear. An excessive feed rate tends to reduce the accuracy of the hole dimensions and the quality of the finish. In reaming nickel alloys, sufficient stock must be removed so that non-work-hardened or non-glazed material is being cut. Good starting points for stock removal are 0.010 inch (0.254 mm) on a 1/4-inch (6-mm) hole, 0.015 inch (0.381 mm) on a 1/2-inch (13-mm) hole and up to 0.025 inch (0.635 mm) on a 1 1/2-inch (40-mm) hole.

Reamers must be kept sharp at all times; those which have been honed will produce smoother surfaces and will last longer between grinds.

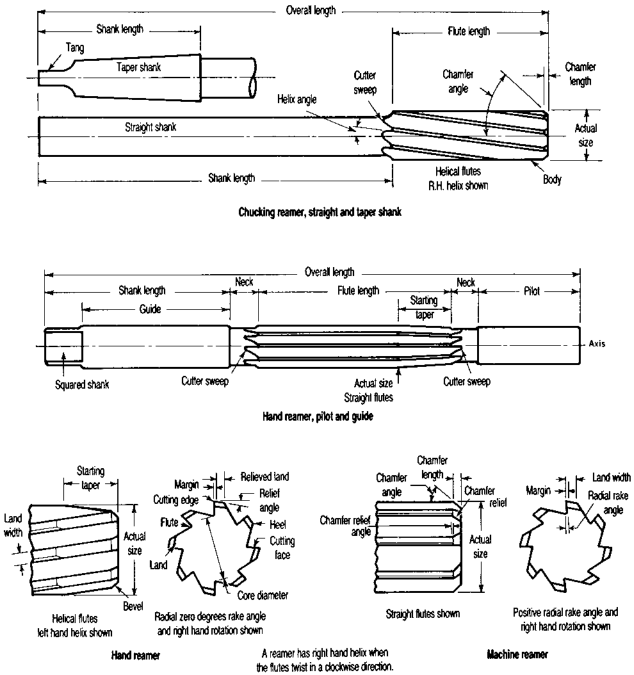


Figure 19. Fluted reamer nomenclature.

A flat reamer is shown in Figure 20 and a built-up reamer head in Figure 21. These tools are used for reaming holes 1 1/2 inches (40 mm) in diameter or larger. Blind holes are reamed with flat reamers, and open or recessed holes with built-up reamers. A reaming allowance of 1/16 to 1/8 inch (1.6 to 3.2 mm) on diameter should be provided for both types. These reamers are mounted on a rigid bar with a bearing support between the work and the lathe tailstock or drilling attachment.

Conventional fluted reamers, flat solid reamers and insert tools for built-up reamers are made of high-speed steel.The molybdenum-bearing grades (Types M-2 and M-10) are preferred because of their toughness. Composite tools having steel shanks tipped with cemented carbide are also used for all types of reamers and are recommended for Group D-2 alloys. Types C-2 and C-6 carbides give good results. Sulfurized or chlorinated oil should be used as lubricant.

Since flat and built-up reamers constitute a specialized area of finishing inside diameters, cutting speeds and feeds must be developed for the job. Speeds of about 67% of those listed for turning with similar tool material and feeds of 0.008 to 0.010 inch (0.203 to 0.254 mm) per revolution should be used as a starting point.

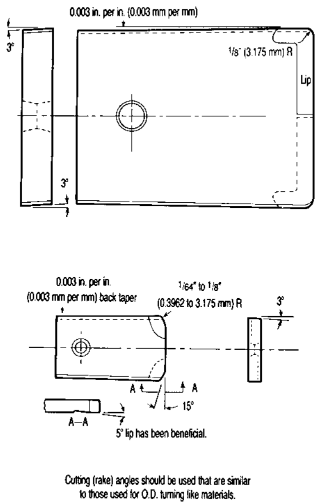


Figure 20. Flat reamers.

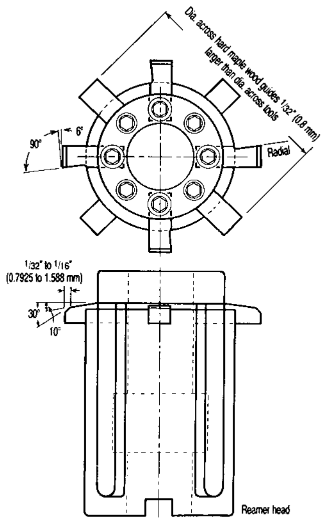


Figure 21. Built-up reamer.

Planing and Shaping

The tools used for planing and shaping are similar to lathe tools, and the cutting actions are almost identical. High-speed steels (types M-2, M-10 and M-34) are preferred.

A suitable tool for rough planing is shown in Figure 22. The top rake angle is the most important; it must be extremely positive to achieve good cutting action. The optimum chip, resulting from a suitable combination of side cutting edge angle and rake angle, is a small curl that curves over before the tool and breaks upon hitting the work.

The gooseneck type of planer tool shown in Figure 23 should be used for finishing. Its spring action effects smooth cuts. It is important that the cutting edge of a gooseneck tool be located behind the center line of the clapper box pin so that the tool will spring away from the cut and not dig in.

Heavy sections may be parted on a planer with the aid of a gooseneck finishing tool such as that shown in Figure 24. Only light cuts, 0.005–0.010 inch (0.127–0.254 mm) per stroke, may be taken. Continuous soluble oil lubrication should be provided.

Cutting oils are not essential for roughing, but sulfurized oil should be applied to the workpiece for smooth finishing cuts. Table 12 lists feeds, cuts and speeds which are used for planing. Speeds are generally 80–85% of those used for lathe cutting.

Shaping operations are similar to planing. A typical tool is shown in Figure 25.

Table 12—Conditions for Planing

	Roughing						Finishing						Parting			
	Depth of Cut		Feed		Table Surface Speed		Depth of Cut		Feed		Table Surface Speed		Feed		Table Surface Speed	
Alloy Group	in.	mm	in.	mm	fpm	m/min.	in.	mm	in.	mm	fpm	m/min.	in.	mm	fpm	m/min.
A	5/8	16	0.050	1.27	50–60	15 –18	0.010	0.254	0.250	6.35	50	15	0.005–0.010	0.127–0.254	50	15
B	5/8	16	0.050	1.27	40–50	12 –15	0.010	0.254	0.250	6.35	40	12	0.005–0.010	0.127–0.254	40	12
C	3/8	10	0.050	1.27	15–20	5 - 6	0.010	0.254	0.250	6.35	15	5	0.005–0.010	0.127–0.254	15	5
D-1	3/8	10	0.050	1.27	20–30	6 - 9	0.010	0.254	0.250	6.35	20	6	0.005–0.010	0.127–0.254	20	6
D-2—Unaged	3/8	10	0.040	1.02	5–10	1.5- 3	0.010	0.254	0.250	6.35	5	1.5	0.005–0.010	0.127–0.254	5	1.5
D-2—Aged	3/8	10	0.040	1.02	5–10	1.5- 3	0.010	0.254	0.250	6.35	5	1.5	0.005–0.010	0.127–0.254	5	1.5

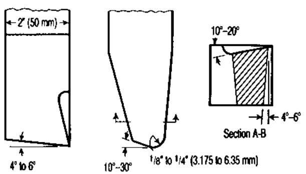


Figure 22. Rough planing tool.

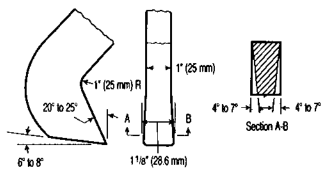


Figure 23. Gooseneck finishing planer tool.

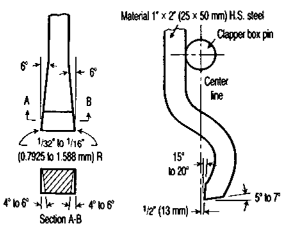


Figure 24. Gooseneck finishing planing tool for heavy sections.

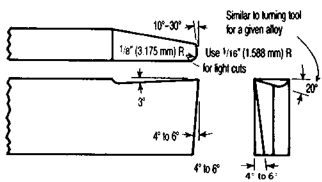


Figure 25. Shaping Tool.

Tapping

The most important factor in tapping is the selection of the proper drill size. The standard tap drill selection tables, in use for many years, are based on 75% thread engagement and were established through experience on low-strength materials such as brass. Modern high-strength materials, however, provide adequate holding strength with lower percentages of thread engagement. For most requirements, 55% is sufficient and more than 60% is seldom required. This is particularly true for holes tapped to a depth of 1 1/2 times the bolt diameter. Thread strength tests show that any increase in thread height above 60% for the tapped member does not increase the static strength of a threading fastening. In general, the bolt will break at 55% engagement.

Decreasing the thread engagement decreases the torque necessary to drive the tap and markedly decreases tap breakage. As a general rule, torque is doubled when thread height is increased from 60 to 72% and tripled when thread height is raised to 80%.

Suggested percentages of thread height are given in Table 13. Tap drill selection tables appear in the Appendix.

For most applications, standard high-speed-steel 4-flute taps are recommended. These standard taps are readily obtained with a 7° hook angle (see Figure 26). The alloys of Group D-2 are best tapped with a series of taps consisting of standard taps modified in diameter so that each successive tap increases the thread diameter proportionately.

Table 13—Tapping

Alloy Group	Surface Speed fpm	Surface Speed m/min.	Recommended Thread Engagement, %
A	15–25	5 –8	60
B	15–25	5 –8	60
C	10–15	3 –5	55
D-1	10–15	3 –5	55
D-2—Unaged	5–10	1.5–3	50
D-2—Aged	5–10	1.5–3	50

Tapping speeds are shown in Table 13. The age-hardenable alloys should be tapped in the unaged condition whenever possible. Ample lubricant is essential for both hand and machine tapping, with liquid chlorinated wax preferred.

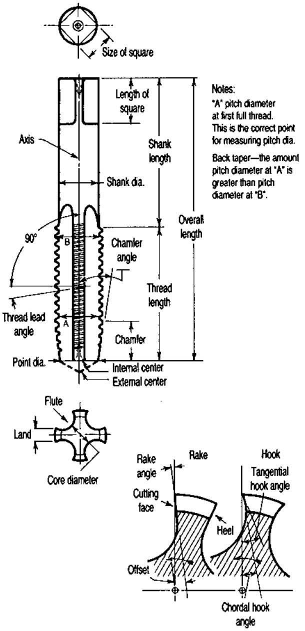


Figure 26. Standard high-speed-steel taps.

Cutting and Sawing

Hacksawing

Hand and power hacksaws are suitable for cutting-off operations involving the alloys of Groups A, B, C and D-1. Alloys of Group D-2 are not readily cut by these tools.

Hand hacksaw blades should be made of high-speed steel. Blades with 14 to 18 (raker-set) teeth per inch (6 to 7 teeth per cm) are used for general work. Blades with wave-set teeth having 24 to 32 teeth per inch (9 to 13 teeth per cm) are used for sawing thin-wall tube.

Power hacksaws may be operated at 90 strokes per minute for the alloys of Groups A and B and about 60 strokes a minute for Groups C and D-1. High-speed-steel blades give satisfactory service. Heavy-duty power hacksaw blades with 6 to 10 (raker-set) teeth per inch (2 to 4 teeth per cm) should be used for cutting bar stock. The same type of blades with 14 to 18 raker or wave-set teeth per inch (6 to 7 teeth per cm) is suitable for cutting off tube with 1/16-inch (1.6-mm) or heavier wall. Tube with less than 1/16-inch (1.6-mm) wall thickness is not generally cut off on power hacksaws, but if necessary a blade with at least 18 (wave-set) teeth per inch (7 teeth per cm) should be used.

The work should be kept flooded with a water-soluble or sulfurized cutting oil.

Circular Sawing

The best method for cold-cutting heavy sections (forgings, ingots, blooms, etc.) of the alloys of Groups A, B, C and D-1 is by means of circular saws with insert teeth. A typical installation uses a 44-inch (1100-mm) diameter blade with 56 inserted high-speed-steel teeth. The teeth are ground and set into the blade to give a 15° rake angle. Square-nosed and round-nosed teeth alternate, with the round-nose teeth projecting about 1/16 inch (1.6 mm) beyond the cutting edge of the square-nosed teeth. This saw cuts alloys of Groups A and B at a speed of 50 feet 15 m) per minute with a 1/3-inch-per-minute (8.5-mm-per-minute) feed and with the aid of a water-soluble cutting oil. Alloys of Groups C and D-1 are cut at 25 feet (7.5 m) per minute with the same feed but with sulfurized oil as a cutting fluid.

Band Sawing

Band sawing can be used for cutting off all the alloys although it is not recommended for Group D-2 alloys of thick section. High-speed-steel saws with flexible backs and in the "A" temper* are recommended. Raker-set teeth are suggested for sawing all forms of material other than light-gauge sheet and thin-wall tube. Saws with wave-set teeth are best for sawing thin sections.

The speeds shown in Table 14 may be used as a guide for establishing cutting rates. Medium feed-

Table 14—Conditions for Band Sawing

Alloy Group	Work Thickness		Teeth		Surface Velocity	
	in.	mm	per Inch	per cm	fpm	m/min.
A	1/16	1.6	14	6	105	32
	1/4	6.4	10	4	75	23
	1	25	8	3	50	15
	3	76	6	2	50	15
B	1/16	1.6	18	7	125	38
	1/4	6.4	14	6	75	23
	1	25	10	4	50	15
	3	76	8	3	50	15
C	1/16	1.6	14	6	90	27
	1/4	6.4	12	5	75	23
	1	25	10	4	50	15
	3	76	8	3	50	15
D-1	1/16	1.6	18	7	75	23
	1/4	6.4	12	5	40	12
	1	25	10	4	30	9
	3	76	8	3	30	9
INCONEL alloy X-750 (cold-rolled, annealed)	.093	2.36	32	13	60	18

*as tempered by the saw manufacturer

ing pressures should be used. The saw should constantly bite into the work; otherwise the blade will work harden the material.

The blade and workpiece should be flooded with a soluble oil. A sulfurized or sulfochlorinated oil may be brushed on the saw teeth to prevent chip welding.

Friction Sawing

Nickel alloys can be readily cut by friction sawing. Irregular as well as straight cuts may be made in material up to 1 inch (25 mm) in thickness. Material under 1/2 inch (13 mm) in thickness may be fed into the saw manually; hydraulic or power feeds should be used for material 1/2 to 1 inch (13 to 25 mm) thick. If it is necessary to use manual feed for the thicker materials, the workpiece should be rocked up and down slowly to expose a smaller surface to the saw.

The mechanism of friction sawing depends on the heat developed between the saw and the workpiece; consequently lubricants are not used for this type of cutting. A new saw blade is not required for friction sawing; the saw becomes efficient only after its teeth become blunt and create the necessary frictional heat to soften the material.

"A" temper saw blades with raker-set teeth should be used. The teeth may be 10, 14 or 18 pitch (4, 6 or 7 teeth per cm), the 10-pitch (4 teeth per cm) saw being used for the thicker materials and the 18-pitch (7 teeth per cm) saw for the thinnest stock. Generally 10-pitch (4 teeth per cm) saws are used for material 5/8 inch (16 mm) thick and heavier. For straight cutting, a 1-inch (25-mm) wide blade is most desirable for all thicknesses. However, for both straight and contour cutting, a good practice is to use a blade that is at least 1/8 inch (3.2 mm) wider than the thickness of the material to be cut. Saw blades less than 1/2 inch (13 mm) wide should be used only to produce contours having radii under 3 inches (75 mm).

Representative saw speeds are:
- 5,000 ft/min. (1500 m/min.) for material 1/32 inch (0.8 mm) thick
- 9,000 ft/min. (2700 m/min.) for material 3/16 inch (5 mm) thick
- 11,000 ft/min. (3350 m/min.) for material 1/2 inch (13 mm) thick
- 15,000 ft/min. (4600 m/min.) for material 1 inch (25 mm) thick

Table 15 lists friction sawing data, developed by several equipment manufacturers, which may be used for guidance.

Lathe Cut-Off

Rounds larger than 7/8 inch (22 mm) may be cut off more rapidly on cut-off lathes than by hacksawing. These lathes usually have two high-speed-steel blades, 3/16- to 1/4-inch (4.76- to 6.35-mm) thick, which should be set in the tool holder so as to give a positive rake. One tool should have a square nose, the other a rounded nose. Operating speeds are 50 to 60 feet per minute (15 to 18 metres per minute) for alloys of Groups A and B and 30 to 35 feet per minute (9 to 11 metres per minute) for those of Groups C and D-1. Water-soluble cutting oil is an adequate lubricant.

Abrasive Cut-Off

All alloys can be cut off with abrasive wheels. For dry cutting small sections, up to 1 inch (25 mm), aluminum oxide resinoid-bonded wheels such as types A301-R6-B4A and A60-Q8B are satisfactory.

Wet cutting is preferred for sections over 1 inch (25 mm) thick and for Groups D-1 and D-2 alloys in all thicknesses. Aluminum oxide rubber-bonded wheels such as type XA602-M-RA are recommended. Water with a rust inhibitor is a satisfactory coolant. Surface speeds should be approximately 5000 to 5500 fpm (1525 to 1675 m/min.), and feeds the maximum permitted by machine capability.

Table 15—Conditions for Friction Sawing, 10-pitch (4 teeth per cm), raker-set blade

Alloy	Work Thickness		Blade Width		Blade Surface Speed		Approximate Lineal Cutting Rate	
	in.	mm	in.	mm	fpm	m/min.	fpm	m/min.
MONEL alloy 400	3/16	5	1	25	9,000	2,700	43	13.1
MONEL alloy K-500[a]	3/16	5	1	25	10,000	3,050	43	13.1
INCONEL alloy 600[a]	3/16	5	1	25	9,000	2,700	72	21.9
INCONEL alloy X-750[a]	1/8	3	5/8	16	9,500	2,900	31	9.4

[a] Annealed condition.

Milling

The essential requirements of milling are high accuracy and smooth finish. In order to achieve these it is imperative to have sharp tools and rigid machines and fixtures.

High-speed-steel cutters (M-2 and M-10) are most suitable for milling because of the interrupted cutting action generally involved.

Recommended feeds and speeds are shown in Table 16. Too light a feed, approximating a "rubbing" condition, will cause a seriously work-hardened layer. Because rubbing action at the beginning of the cut is avoided by climb milling, this technique is preferred to up milling. In addition, the downward motion of the cut assists rigidity and diminishes the tendency to chatter. The disadvantage of climb milling is the necessity for positive control of backlash in the table drive. Face milling is preferable to slab milling also because of reduced tendencies toward work hardening and chattering.

Table 16—Recommended Milling Procedures

Alloy Group	Operation	Surface Cutting Speed		Feed per Tooth	
		fpm	m/min.	in.	mm
A	Helical Milling	80–100	24 –30	0.003	0.076
	Face Milling			0.004	0.102
	Side Milling			0.002	0.051
	End Milling			0.002	0.051
	Slotting			0.002	0.051
	Sawing			0.002	0.051
B	Helical Milling	60–80	18 –24	0.007	0.178
	Face Milling			0.008	0.203
	Side Milling			0.005	0.127
	End Milling			0.004	0.102
	Slotting			0.005	0.127
	Sawing			0.002	0.051
C	Helical Milling	30–40	9 –12	0.005	0.127
	Face Milling			0.006	0.152
	Side Milling			0.004	0.102
	End Milling			0.003	0.076
	Slotting			0.004	0.102
	Sawing			0.002	0.051
D-1	Helical Milling	25–35	8 –11	0.005	0.127
	Face Milling			0.006	0.152
	Side Milling			0.004	0.102
	End Milling			0.003	0.076
	Slotting			0.004	0.102
	Sawing			0.002	0.051
D-2—Unaged	Helical Milling	10–20	3 - 6	0.003	0.076
	Face Milling			0.004	0.102
	Side Milling			0.003	0.076
	End Milling			0.002	0.051
	Slotting			0.003	0.076
	Sawing			0.001	0.025
D-2—Aged	Helical Milling	5–15	1.5- 4.5	0.003	0.076
	Face Milling			0.004	0.102
	Side Milling			0.003	0.076
	End Milling			0.002	0.051
	Slotting			0.003	0.076
	Sawing			0.001	0.025

Chip problems in milling are the same as those for turning. The milling chips tend to curl, and sufficient space must be available between cutter teeth to accommodate the curls. Standard milling cutters provide adequate chip clearance.

Heavy-duty milling cutters having 12° positive radial rake and 45° axial rake are preferred for rough milling all alloys except those of Group D-2. Light-duty cutters with 12° positive radial rake and 18° axial rake (helical flutes) are best for the high-strength alloys of this group. They require low surface cutting speeds, 10–20 fpm (3–6 m/min.) and light chip loads. The light-duty cutters have more teeth than the heavy-duty type and thus operate at higher cutting rates for the cutting speeds allowed.

Finishing cutters for all alloys should be of the high helix type with 15° positive radial rake and 52° to 65° helical flutes (positive axial rake). Staggered tooth cutters, with alternate teeth of opposite helix, are best for milling grooves. High-speed-steel slitting saws with side chip clearance are recommended for narrow slotting.

Face-milling cutters with inserted teeth of high-speed steel should be designed so that the inserted teeth have positive rake and helix angles. Figure 27 shows a typical tool with cutters set into the head at a positive axial rake angle or helix of 7° and a positive radial rake angle of 15°. Primary relief angles should be 7–8°; secondary relief angles, 12–14° on all cutters except end mills and small diameter cutters. Consult tool handbook for clearance on periphery on small end mills and use the same clearance, in accordance with corresponding diameters, as for alloy steel.

Typical milling tools are shown in Figure 28.

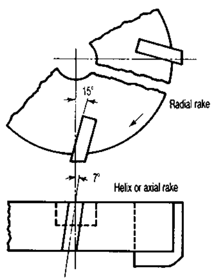
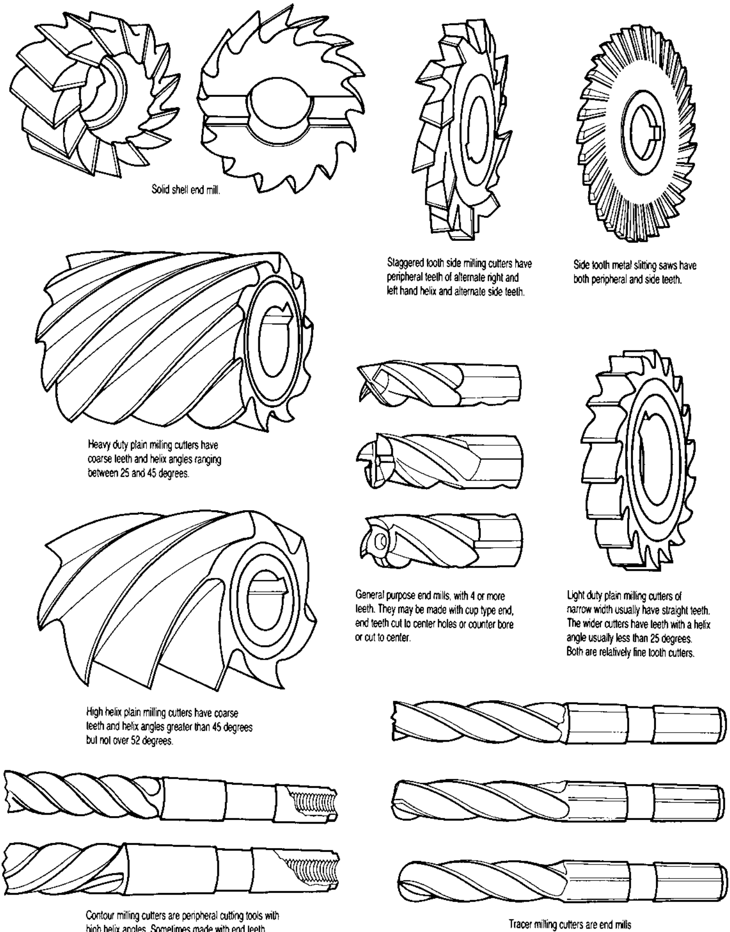


Figure 27. Typical face-milling tool.



Solid shell end mill.

Staggered tooth side milling cutters have peripheral teeth of alternate right and left hand helix and alternate side teeth.

Side tooth metal slitting saws have both peripheral and side teeth.

Heavy duty plain milling cutters have coarse teeth and helix angles ranging between 25 and 45 degrees.

General purpose end mills, with 4 or more teeth. They may be made with cup type end, end teeth cut to center holes or counter bore or cut to center.

Light duty plain milling cutters of narrow width usually have straight teeth. The wider cutters have teeth with a helix angle usually less than 25 degrees. Both are relatively fine tooth cutters.

High helix plain milling cutters have coarse teeth and helix angles greater than 45 degrees but not over 52 degrees.

Contour milling cutters are peripheral cutting tools with high helix angles. Sometimes made with end teeth.

Tracer milling cutters are end mills designed for tracer milling.

Figure 28. Typical milling tools.

Grinding

Methods of grinding the high-nickel materials do not differ greatly from the practices used for steel. When only a small amount of metal must be removed, the finishing operation may be done on a grinding machine, using a rough and then a fine grind.

If an extremely accurate ground finish is required, particularly on material of hard temper, the work should be allowed to cool to room temperature after the final roughing cut or grind. This procedure allows redistribution of internal stresses, and the resulting distortion, if any, can be corrected in the final grinding operation.

For best results, the alloys should be ground wet. A solution of 25 gallons (95 liters) of water and 1 pound (0.5 kg) of sal soda, or a solution of 50 parts of water to 1 part of soluble oil, is a suitable grinding lubricant for operations other than crush form and thread grinding. A good grinding oil is the best lubricant for crush form and thread grinding. Sodium chromate may be added to sal soda solutions to inhibit rusting of the equipment.

In general, silicon carbide grinding wheels give best results on alloys of Groups A, B, D-1, and E; aluminum oxide wheels are best for alloys of Groups C and D-2. Grinding pressures should be great enough to cause slight wheel breakdown. Because of the many variables encountered in grinding, the wheel manufacturer should be consulted for information on specific applications.

Surface Grinding

Coarse-grit (46–60) aluminum oxide wheels produce the best finishes in surface grinding. To avoid warping during grinding, the workpiece should be in the stress-equalized condition. Low wheel contact and low pressure help prevent distortion during grinding, especially with annealed material. Reciprocating tables are preferred to rotary tables. Reciprocating tables reduce wheel contact, generate less heat, and cause less distortion of the work.

Centerless Grinding

Centerless grinding should be done with a wheel having a face that will break down during operation and prevent the workpiece from becoming out of round. The breakdown of the wheel face depends on the diameter of the work, infeed per pass, and the angle and speed of the regulating wheel. Diamond-dressed wheels are more prone to cause ovality of the work than are wheels dressed on a Ross dressing device. This dresser produces a sharp wheel face similar to one that has been crushed-dressed. By taking light cuts on the material, finish grinding can be done without redressing the wheel after the roughing operation.

Crush Form Grinding

Vitrified-bonded, medium-grade aluminum oxide wheels having medium-to-open structures produce good results in crush form grinding. A high-grade grinding oil is recommended and should be continuously filtered through all operations.

Grinding with Abrasive Belts

Abrasive belts (cloth belts coated with aluminum oxide) can be used for finishing Group D alloys. One procedure used for abrasive-belt finishing of precision-forged airfoil turbine blades consists of rough grinding with 80 grit followed by semi-finishing with 120–150 grit and final finishing with 180–220 grit. Rough grinding can be done dry or with a lubricant. A machine oil of high flowing characteristic is suitable for rough grinding. Semi-finish and finish grinding is done with a lubricant such as cottonseed oil. The addition of kerosene imparts a high flowing characteristic to the oil.

Honing

Honing is done with aluminum oxide vitrified-bonded honing stones of medium-to-soft grade. The honing stone must have uniform breakdown characteristics. Ample coolant must be supplied; proprietary honing oils, either as-supplied or diluted by 2 to 3 parts of kerosene, are recommended. A mixture of 50% oleic acid, 35% kerosene, and 15% turpentine is also suitable.

Surface speeds for rotation of the hone are between 150 and 250 fpm (46 and 76 m/min.). Reciprocation surface speeds are between 35 and 50 fpm (11 and 15 m/min.). The lower speeds are used for roughing and the higher speeds for finishing. Honing pressure should be about 450 psi (3.1 MPa). The manufacturers of honing stones should be consulted for detailed recommendations on specific problems.

Automatic Machining

MONEL alloy R-405 was specifically developed for good machinability. It is recommended for use with automatic screw machines. Other alloys in Groups A, B, C and D-1 may be machined on automatics, but the lower speeds required are generally not possible with this type of equipment. Recommended cutting speeds and feeds for MONEL alloy R-405 are shown in Table 17 and the types of tools are shown in Figure 29.

MONEL alloy R-405 combines the toughness, strength, and corrosion resistance of MONEL alloy 400 with excellent machinability. This "free-machining" characteristic is obtained by careful adjustment of minor alloying constitutents, including the controlled addition of a small amount of sulfur. The resulting nickel-copper sulfides in the material act as chip breakers. Because of these inclusions the surface finish of the alloy is not as good as that of MONEL alloy 400. Results obtained in

Table 17—Recommended Cutting Speeds and Feeds for MONEL alloy R-405

Operation	Surface Speed		Feed	
	fpm	m/min.	ipr	mm/rev.
Turning[a]	140–160	43–49	0.003 –0.005	0.076 –0.127
Forming[a]	140–160	43–49	0.0004–0.001	0.0102–0.025
Drilling	60–80	18–24	0.001 –0.005	0.025 –0.127
Reaming	30–45	9–14	0.003 –0.012	0.076 –0.305
Tapping	30–40	9–12	-	-
Threading	30–40	9–12	-	-
Cut-off	140–160	43–49	0.0005–0.001	0.0127–0.025

[a] For a single-spindle screw machine handling stock 1 inch (25 mm) in diameter and under, using high-speed-steel tooling. For a multiple-spindle machine or a lathe handling stock over 1 inch (25 mm) in diameter, use surface speeds of 90 to 125 fpm (27 to 38 m/min.).

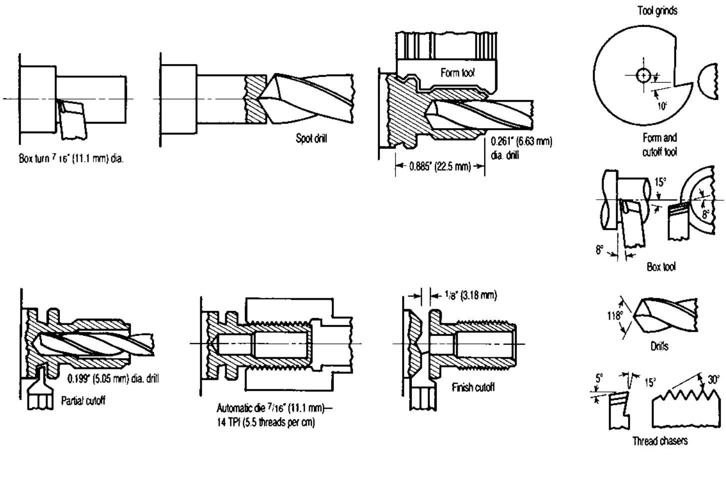


Figure 29. Tools used in automatic screw machining of 5/8-in. (16 mm) diameter MONEL alloy R-405. (See Table 18, Test 1.)

actual production runs in commercial screw machines are illustrated in Table 18.

The preferred materials for automatic machining tools are molybdenum-containing high-speed steels (Type M-2, M-10 or M-36). Cemented carbide tools are successfully used for finish turning and for shaving (Figure 30). The following discussion is specifically applicable to MONEL alloy R-405. Drilling, tapping and other operations not described should be carried out in accordance with recommendations contained in earlier pages of this publication and in accord with data in Table 17.

Table 18—Automatic Screw Machining of MONEL alloy R-405

	TEST I[a]								TEST II				TEST III			
	Size of Material—5/8 in. (16 mm) diameter Tooling—High-speed steel Lubricant—Sulfurized chlorinated cutting oil								Size of Material—5/8 in. (16 mm) hexagon Tooling—High-speed steel Lubricant—Sulfurized mineral-base sperm oil				Size of Material—5/8 in. (16 mm) hexagon Tooling—High-speed steel Lubricant—Sulfurized mineral-base sperm oil			
	Surface Speed		Feed		Surface Finish				Surface Speed		Feed		Surface Speed		Feed	
					First Piece		Last Piece									
Operation	fpm	m/min.	ipr	mm/rev.	microinch	µm	microinch	µm	fpm	m/min.	ipr	mm/rev.	fpm	m/min.	ipr	mm/rev.
Box Turning	143	44	0.004	0.102	—	—	—	—	—	—	—	—	—	—	—	—
Spot Drilling	65	20	0.0035	0.0889	—	—	—	—	—	—	—	—	—	—	—	—
Forming	143	44	0.001	0.025	25–45	0.64–1.1	100	2.54	161	49.0	0.0007	0.0178	176	53.6	0.006	0.152
Drilling																
0.261 in. (6.63 mm) diam.	60	18	0.003	0.076	8–20	0.2 –0.51	30–60	0.76–1.5	—	—	—	—	—	—	—	—
0.199 in. (5.05 mm) diam.	46	14	0.0025	0.0635	45	1.1	—	—	—	—	—	—	—	—	—	—
3/8 in. (9.53 mm) diam.	—	—	—	—	—	—	—	—	84	25.6	0.004	0.102	—	—	—	—
9/32 in. (7.14 mm) diam.	—	—	—	—	—	—	—	—	62	18.9	0.0035	0.0889	—	—	—	—
1/2 in. (12.7 mm) diam.	—	—	—	—	—	—	—	—	—	—	—	—	112	34.1	0.005	0.127
3/8 in. (9.53 mm) diam.	—	—	—	—	—	—	—	—	—	—	—	—	84	25.6	0.0045	0.1143
Reaming																
3/8 to 1/2 in. (9.53 to 12.7 mm) step	—	—	—	—	—	—	—	—	21	6.4	0.0075	0.1905	—	—	—	—
17/32 in. (13.5 mm) diam.	—	—	—	—	—	—	—	—	—	—	—	—	38	11.6	0.018	0.457
Tapping																
9/16 in. (14.3 mm)—20 TPI (8 threads per cm)	—	—	—	—	—	—	—	—	—	—	—	—	41	12.5	0.050	1.27
Threading																
7/16 in. (11.1 mm)—14 TPI (6 threads per cm)	33	10	0.071	1.803	Smooth[b]	Smooth[b]	Smooth[b]	Smooth[b]	—	—	—	—	—	—	—	—
9/16 in. (14.3 mm)—20 TPI (8 threads per cm)	—	—	—	—	—	—	—	—	25	7.6	0.050	1.27	—	—	—	—
Cut-off	143	44	0.001	0.025	40–80	1.0–2.0	30	0.76	100	30.5	0.0008	0.0203	140	42.7	0.008	0.0203
Results	575 parts were produced in 75 seconds machine time per piece. Tools were not excessively worn after 11.8 hours.								495 parts were produced in 60 seconds machine time per piece. Tools were not excessively dulled after 8.2 hours. Surface finish was equal to or better than that in Test I.				543 parts were produced in 45 seconds machine time per piece. Tools required regrinding after 6.7 hours. Surface finish was equal to or better than that in Test I.			

[a] Tools used are shown in Figure 29. [b] Visual examination.

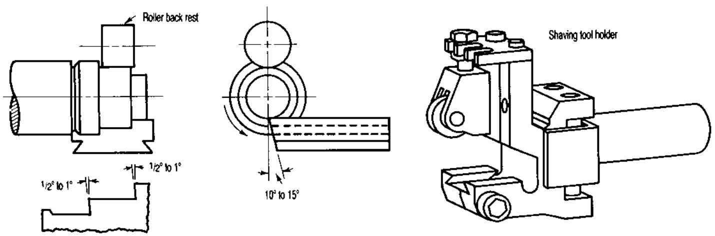


Figure 30. Dovetail shaving tool.

Form Tools

Form tools are extensively used for automatic machining. The form tools must be sturdy, and a rigid set-up must be maintained in order to achieve efficient operation and adequate tool life.

Recommended form tool grinds are shown in Figures 31–34. The cutting edge of the form tool is set in line with the centerline of the work. Form tools designed with positive top rake angles are more efficient than form tools with 0° top rake angles. Sharp corners should be eliminated because they lead to early tool breakdown. Figure 33 shows a modified form tool design that will aid in minimizing the drag encountered on a conventional ground circular or dovetail form tool because of its lack of side clearance. The excessive stock left in the form by the modified tool can be removed by a typical cut-off or parting tool. The flat form tool shown in Figure 34 is used for forms of simple design and where there is sufficient side clearance for the cutting edges.

The recommended maximum width for form tools ranges between 1½ to 2 times the minimum diameter that is to be formed. The use of wider form tools is quite likely to result in chatter and consequent short tool life. When a job requires the use of a wider form tool it is best to do the forming in two separate operations using tools whose combined width is equal to the width of the specified form.

The in-cutting or plunge-cutting action of forming tools is less desirable than longitudinal cutting as with single-point tools. Consideration should be given to first roughing down the section to be formed with a longitudinal cut so that the remaining amount of stock to be removed in forming is held to the minimum. If longitudinal cutting cannot be done, the use of separate tools for rough and finish forming will be more economical than use of a single forming tool.

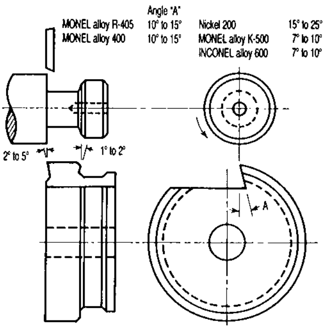


Figure 31. Circular form tool.

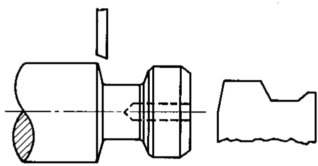


Figure 33. Modified form tool for minimizing drag.

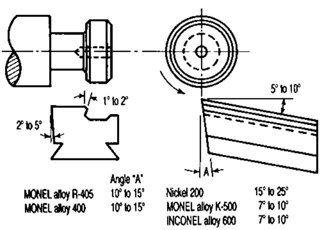


Figure 32. Dovetail form tool.

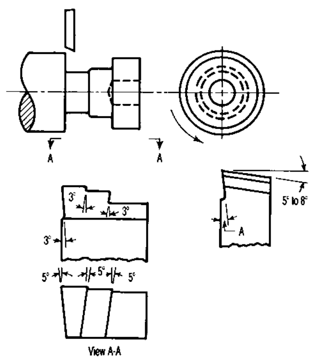


Figure 34. Flat form tool.

Box Tools

Box tools having V-shaped or roller work supports are used for roughing and finishing cuts. Box tools are preferred to balance turning tools because they provide more rigidity. A suitable tool grind for a box tool cutter is shown in Figure 35.

Balance Turning Tools

Balance turning tools are primarily used for roughing cuts. They offer no means for supporting the work and should not be used for turning long slender parts. The blades are held in the tool holder at a fixed angle, which must be considered when grinding the tools. Figure 36 shows the cutting edge angles that are suitable for a balance turning tool and illustrates a grind for reducing the front relief when a heavier cutting edge is desired.

Cut-Off Tools

Figures 37 and 38 illustrate typical circular- and straight-blade automatic-screw-machine cut-off tools. Circular cut-off tools are suitable, but straight cut-off tools afford greater clearance below the cutting edges and are preferred.

The blade thickness should be greater than is used for cutting equivalent sections of carbon steel. A next-heavier-size blade than would be used for the same diameter of carbon steel is recommended.

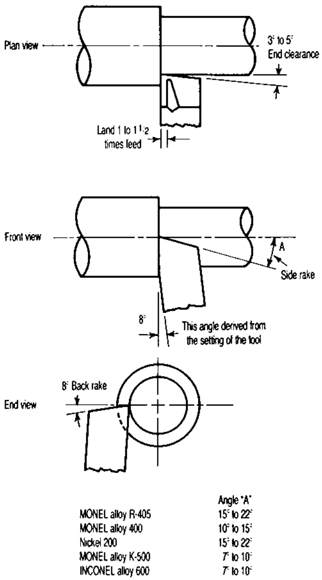


Figure 35. Box tool cutter.

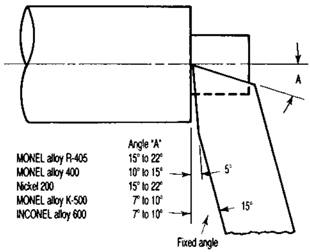


Figure 36. Balance turning tool.

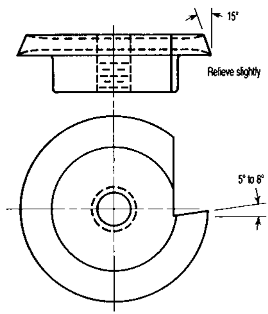


Figure 37. Circular automatic cut-off tool.

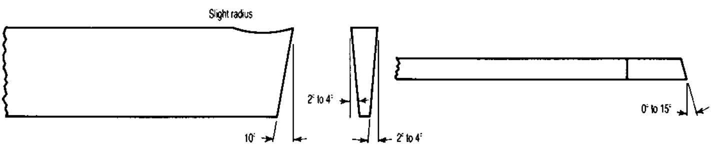


Figure 38. Straight-blade automatic cut-off tool.

Chemical and Electrical Machining

Chemical and electrical machining can be economical or perform operations not easily accomplished by conventional machine tools. A few of these techniques are briefly described below. Because of rapidly expanding and changing technology, vendors of the equipment for these processes should be consulted for detailed recommendations on specific problems.

Electrochemical machining (ECM) is essentially electrolytic deplating in which the "cutting tool" is a shaped cathode, usually of copper, and the workpiece is made anodic. Metal-removal rates are not affected by the mechanical properties of the workpiece but rather depend upon the gram equivalent weight of the alloy (Faraday's Law) and the output of the equipment in terms of current density at the electrodes. Plating of the cathode is avoided by precipitation of the dissolved metal in the form of insoluble salts. Electrolytes used are generally proprietary in composition but are basically neutral salt solutions (sodium chloride, potassium nitrate) or dilute acid (sulfuric).

The resistance of nickel alloys to chemical attack does not seem to retard electrochemical solution. Some differences, however, will occur in processing the various alloy families because of differences in their current-carrying capacities. The nickel and nickel-copper alloys, for instance, have fair-to-good electrical conductivities whereas the chromium-bearing materials are essentially resistance alloys. Compensations will have to be made for greater power loss, voltage drop and electrolyte heating with the latter alloys. Since mechanical properties do not affect ECM, alloys can be shaped in the fully cold-worked and age-hardened condition as easily as in the annealed condition. This is particularly advantageous with the D-2 Group of alloys and permits operations such as small-diameter deep hole drilling not considered feasible by conventional methods.

Electrolytic grinding is a specialized form of ECM in which the cathode is a current-conducting grinding wheel, usually a metal wheel impregnated with abrasive particles or diamond dust. About 90% of metal removal is accomplished by electrolytic solution and about 10% by abrasion so wheel life is relatively long. The principal effect of the abrasive action is to grind off any passive film, especially on chromium-bearing alloys, that might result from oxidizing acid or salt solutions.

Electrical discharge machining, or spark erosion, originally developed for removal of broken taps and later adapted for shaping of carbides and hard, relatively brittle tool steels, has been refined for machining of all types of current-carrying materials. Little is known about the actual mechanism of metal removal, but the most popular theory is that the metal is melted by the thermal energy of the spark. This would explain its effectiveness on tough, ductile materials and why the process is not influenced greatly by the mechanical properties and heat treatment of the workpiece. Experiments have indicated that the nonchromium alloys in the D-2 Group require approximately 75% more power than tool and die steels, and that metal removal rates approximately 12–14 cubic inches/hour (200–230 cubic centimeters/hour) can be achieved on Group C material at higher power settings. With proper control, very fine finishes and close dimensional tolerances can be obtained.

Chemical milling is controlled etching on a production basis, which has been used to advantage in the aerospace field. By use of proprietary solutions, etching rates can be carefully controlled to give close tolerances and fairly smooth surfaces. The process is rather limited in application but is useful in milling large areas at one time, particularly from material such as sheet stock too fragile to mill conventionally. Thin material can be subtly contoured or tapered by chemical milling.

Being generally resistant to chemical attack, the nickel alloys require strong acids for metal removal at practical rates. These acids can cause intergranular attack or severe pitting if not properly used, especially on the chromium-bearing alloys. Quality control should include a sample metallographic examination of the finished surface. Even under the best conditions, chemical milling will lower fatigue life slightly, probably because work-hardened surface layers have been removed and because incipient grain-boundary attack has occurred.

Final surface finish will depend on grain size, starting surface finish of the workpiece, and composition of the solution. As in ECM, mechanical properties have no direct effect on chemical milling.

One of the major problems to be overcome in chemical milling the chromium-bearing alloys is the prevention of preferential attack at grain boundaries. When nickel-chromium alloys are heated in the temperature range of 1000° to 1400°F (540° to 760°C), chromium carbide precipitates in the grain boundaries, leaving the area near the boundary less rich in chromium and more susceptible to intergranular attack. Thus, the age-hardenable alloys should be in the solution-treated or the unaged condition before milling; and if it becomes necessary to mill after aging, compensation must be made in the etching solution.

Electron-beam machining and plasma-arc cutting are two similar techniques which remove metal by localized melting and vaporization with high-temperature, high-velocity beams of electrons or ionized gas.

Briefly, both methods employ an electron beam generated by a heated tungsten electrode. Electron-beam machining is performed in a vacuum and depends on the energy of the electrons only, whereas the plasma-arc cutting process can function in air with the electron beam being used to ionize a stream of inert gas such as argon into plasma.

It appears that electron-beam machining will be confined to precision cutting and drilling of thin-gage material and similar operations. However, plasma-arc torches have been found capable of cutting nickel alloys in thick sections at very high speeds and are proving superior to all other methods in the rough cutting of plate and sheet. Both processes develop temperatures considerably above the melting points of the alloys and are only slightly affected by alloy composition and heat treatment. The thermal mass of the "flame" or arc can be kept quite small so that the heat-affected zone is reduced to a minimum.

Ultrasonic machining is a form of abrasive machining in which the abrasive particles, suspended in a liquid medium, are propelled linearly into the workpiece by a vibrating tool. Vibration is achieved with a magnetostrictive transducer, attached to the tool, which converts high-frequency electrical energy into ultrasonic linear motion. The tool never touches the workpiece but causes the abrasive to chip away little pieces of material.

Ultrasonic grinding may improve cutting rates and wheel life in grinding many materials, including the nickel alloys. In this process, vibrations are intentionally introduced into the grinding process through the ultrasonic transducer.

Cold machining involves supercooling of the cutting tool and workpiece by various means to reduce the undesirable effects of the heat generated by machining, such as tool wear, dimensional instability, chip welding, and lubricant staining. Some of the methods used for cooling are refrigerating the workpiece prior to machining, refrigerating the coolant, using a refrigerant such as CO_2 mist as a coolant and cooling by thermoelectric effect, in which the tool bit and workpiece become a thermocouple while in contact.

Appendix

Tap Drill Selection

Guides for tap-drill selection are shown in following Tables 19 and 20 for theoretical thread engagements of 50 to 83¹/₃%. The data for 83¹/₃% engagement are applicable to jobs in a screw-cutting lathe in preparation for chasing internal close-tolerance threads with a single-point tool.

The first step in use of the tables is selection of the desired screw size (left-hand column) and theoretical thread engagement. Consider the block opposite 4–40 and under 70%. The top figure, 0.0893, is the theoretical hole size, in inches. The next two figures show the size of the nearest metric drill in millimeters, 2.25, and in parentheses its equivalent in inches (0.0886). The bottom figure is the number of the American Standard, in this case No. 43 (0.0890 in.). Allowing for the drill cutting oversize, the thread height will probably be about 65%. In cases where no standard drill is shown, another thread height must be selected. As in the example discussed above, under most conditions drills cut slightly larger holes than their nominal size so that thread engagements are slightly less than those shown.

Table 19—Machine-Screw Sizes[a]

Screw Size	Theoretical Thread Engagement (%)						
	83¹/₃	75	70	65	60	55	50
0–80	0.0465 1.18 mm (0.0465) 56 (0.0465)	0.0479 1.21 mm (0.0477) ³/₆₄ (0.0469)	0.0486 1.23 mm (0.0485)	0.0494 1.25 mm (0.0492)	0.0502 1.27 mm (0.0500)	0.0510 1.29 mm (0.0508)	0.0519 1.31 mm (0.0516) 55 (0.0520)
1–64	0.0561 1.43 mm (0.0564) 54 (0.0550)	0.0578 1.47 mm (0.0579)	0.0588 1.49 mm (0.0587)	0.0599 1.50 mm (0.0591) 53 (0.0595)	0.0609 1.55 mm (0.0611)	0.0619	0.0629 1.60 mm (0.0630) ¹/₁₆ (0.0625)
1–72	0.0580 1.47 mm (0.0579)	0.0595 1.50 mm (0.0591) 53 (0.0595)	0.0604	0.0613 1.55 mm (0.0611)	0.0622	0.0631 1.60 mm (0.0630) ¹/₁₆ (0.0625)	0.0604 52 (0.0635)
2–56	0.0667 1.70 mm (0.0670)	0.0686 1.75 mm (0.0689) 51 (0.0670)	0.0698 50 (0.0700)	0.0710 1.80 mm (0.0709)	0.0721 1.85 mm (0.0729)	0.0732 49 (0.0730)	0.0744 1.90 mm (0.0748)
2–64	0.0691 1.75 mm (0.0689)	0.0708 1.80 mm (0.0709) 50 (0.0700)	0.0718	0.0729 1.85 mm (0.0729)	0.0739 49 (0.0730)	0.0749 1.90 mm (0.0748)	0.0759 48 (0.0760)
3–48	0.0764 1.95 mm (0.0768) 48 (0.0760)	0.0788 2.00 mm (0.0787) 47 (0.0785)	0.0801 2.05 mm (0.0807)	0.0815 46 (0.0810)	0.0828 2.10 mm (0.0827) 45 (0.0820)	0.0841 2.15 mm (0.0846)	0.0855 44 (0.0860)
3–56	0.0797 2.00 mm (0.0787)	0.0816 2.05 mm (0.0807) 46 (0.0810)	0.0828 2.10 mm (0.0827) 45 (0.0820)	0.0840 2.15 mm (0.0846)	0.0851	0.0862 44 (0.0860)	0.0874 2.20 mm (0.0866)
4–40	0.0849 2.15 mm (0.0846) 45 (0.0820)	0.0877 2.20 mm (0.0866) 44 (0.0860)	0.0893 2.25 mm (0.0886) 43 (0.0890)	0.0909 2.30 mm (0.0906)	0.0926 2.35 mm (0.0925)	0.0942 2.40 mm (0.0945) ³/₃₂ (0.0938)	0.0958 2.45 mm (0.0965) 41 (0.0960)

Table 19—Machine-Screw Sizes[a] —Continued

Screw Size	Theoretical Thread Engagement (%)						
	83 1/3	75	70	65	60	55	50
4–48	0.0894 43 (0.0890)	0.0918 2.30 mm (0.0906)	0.0931 2.35 mm (0.0925) 42 (0.0935)	0.0945 2.40 mm (0.0945) 3/32 (0.0938)	0.0958 41 (0.0960)	0.0971 2.45 mm (0.0965)	0.0985 2.50 mm (0.0984) 40 (0.0980)
5–40	0.0979 40 (0.0980)	0.1007 2.55 mm (0.1004) 39 (0.0995)	0.1023 2.60 mm (0.1024) 38 (0.1015)	0.1039 37 (0.1040)	0.1056 2.65 mm (0.1044)	0.1072 2.70 mm (0.1063) 36 (0.1065)	0.1088 2.75 mm (0.1083) 35 (0.1100)
5–44	0.1004 2.55 mm (0.1004) 39 (0.0995)	0.1029 2.60 mm (0.1024) 38 (0.1015)	0.1044 2.65 mm (0.1044) 37 (0.1040)	0.1059 2.70 mm (0.1063)	0.1073 36 (0.1065)	0.1087 2.75 mm (0.1083)	0.1102 2.80 mm (0.1102) 35 (0.1100)
6–32	0.1042 2.65 mm (0.1044) 37 (0.1040)	0.1076 36 (0.1065)	0.1096 2.75 mm (0.1082) 7/64 (0.1094)	0.1117 2.85 mm (0.1122) 34 (0.1110)	0.1137 33 (0.1130)	0.1157 2.90 mm (0.1142)	0.1177 32 (0.1160)
6–40	0.1109 2.80 mm (0.1102) 34 (0.1110)	0.1137 2.85 mm (0.1122) 33 (0.1130)	0.1153 2.90 mm (0.1142)	0.1169 2.95 mm (0.1160) 32 (0.1160)	0.1185 3.00 mm (0.1181)	0.1201 31 (0.1200)	0.1218 3.10 mm (0.1220)
8–32	0.1302 3.30 mm (0.1299) 30 (0.1285)	0.1336 3.40 mm (0.1339)	0.1356	0.1377 3.50 mm (0.1378) 29 (0.1360)	0.1397	0.1417 3.60 mm (0.1417) 28 (0.1405)	0.1437 27 (0.1440)
8–36	0.1339 3.40 mm (0.1339)	0.1370 29 (0.1360)	0.1388 3.50 mm (0.1378)	0.1406 28 (0.1405)	0.1424 3.60 mm (0.1417)	0.1442 27 (0.1440)	0.1460 3.70 mm (0.1457) 26 (0.1470)
10–24	0.1449 27 (0.1440)	0.1495 3.80 mm (0.1496) 25 (0.1495)	0.1522 24 (0.1520)	0.1549 23 (0.1540)	0.1576 4.00 mm (0.1575) 22 (0.1570)	0.1602 21 (0.1590)	0.1629 4.10 mm (0.1614) 20 (0.1610)
10–32	0.1562 5/32 (0.1562)	0.1596 21 (0.1590)	0.1616 4.10 mm (0.1614) 20 (0.1610)	0.1637	0.1657 4.20 mm (0.1654)	0.1677 4.25 mm (0.1673) 19 (0.1660)	0.1697 18 (0.1695)
12–24	0.1709 18 (0.1695)	0.1755 17 (0.1730)	0.1782 16 (0.1770)	0.1809 4.60 mm (0.1811) 15 (0.1800)	0.1836 14 (0.1820)	0.1862 4.70 mm (0.1850) 13 (0.1850)	0.1889 4.80 mm (0.1890) 3/16 (0.1875)
12–28	0.1773 4.50 mm (0.1772) 16 (0.1770)	0.1813 4.60 mm (0.1811) 15 (0.1800)	0.1836 14 (0.1820)	0.1859 4.70 mm (0.1850) 13 (0.1850)	0.1882 4.75 mm (0.1870) 3/16 (0.1875)	0.1905 4.80 mm (0.1890) 12 (0.1890)	0.1928 4.90 mm (0.1929) 11 (0.1910)

[a] See text for instructions for use.

Table 20—Coarse- and Fine-Thread Series[a]

Screw Size	Theoretical Thread Engagement (%)						
	83 1/3	**75**	**70**	**65**	**60**	**55**	**50**
1/4–20	0.1959 9 (0.1960)	0.2012 7 (0.2010)	0.2046 5.20 mm (0.2047) 6 (0.2040)	0.2078 5.25 mm (0.2067) 5 (0.2055)	0.2111 4 (0.2090)	0.2143 3 (0.2130)	0.2175 5.50 mm (0.2165)
1/4–28	0.2113 4 (0.2090)	0.2153 3 (0.2130)	0.2176 5.50 mm (0.2165)	0.2199 7/32 (0.2188)	0.2222 2 (0.2210)	0.2245 5.70 mm (0.2244)	0.2268 5.75 mm (0.2264) 1 (0.2280)
5/16–18	0.2524 6.40 mm (0.2520) 1/4 (0.2500)	0.2584 F (0.2570)	0.2620 G (0.2610)	0.2656 6.75 mm (0.2657) 17/64 (0.2656)	0.2692 6.80 mm (0.2677) H (0.2660)	0.2728 6.90 mm (0.2717) I (0.2720)	0.2764 7.00 mm (0.2756) J (0.2770)
5/16–24	0.2674 6.80 mm (0.2677) H (0.2660)	0.2720 6.90 mm (0.2717) I (0.2720)	0.2747 7.00 mm (0.2756)	0.2774 J (0.2770)	0.2801 7.10 mm (0.2795)	0.2827 7.20 mm (0.2835) K (0.2810)	0.2854 7.25 mm (0.2854)
3/8–16	0.3073 7.80 mm (0.3071)	0.3142 8.00 mm (0.3150) 5/16 (0.3125)	0.3182 8.10 mm (0.3189) O (0.3160)	0.3223 8.20 mm (0.3228)	0.3263 8.30 mm (0.3268) P (0.3230)	0.3303 8.40 mm (0.3307) 21/64 (0.3281)	0.3344 8.50 mm (0.3346) Q (0.3320)
3/8–24	0.3299 21/64 (0.3281)	0.3345 8.50 mm (0.3346) Q (0.3320)	0.3372 8.60 mm (0.3386)	0.3399 R (0.3390)	0.3426 8.70 mm (0.3425)	0.3452 8.75 mm (0.3445) 11/32 (0.3438)	0.3479 8.80 mm (0.3465) S (0.3480)
7/16–14	0.3602 23/64 (0.3594)	0.3680 U (0.3680)	0.3726 9.40 mm (0.3701)	0.3772 V (0.3770)	0.3819 9.70 mm (0.3819)	0.3865 W (0.3860)	0.3911 25/64 (0.3906)
7/16–20	0.3834 9.70 mm (0.3819)	0.3888 W (0.3860)	0.3921 25/64 (0.3906)	0.3953 10.00 mm (0.3937)	0.3986 X (0.3970)	0.4018	0.4050 Y (0.4040)
1/2–13	0.4167 10.50 mm (0.4134) Z (0.4130)	0.4251 27/64 (0.4219)	0.4301	0.4351 11.00 mm (0.4331)	0.4401 7/16 (0.4375)	0.4450	0.4500 11.50 mm (0.4528) 29/64 (0.4531)
1/2–20	0.4459	0.4513 11.50 mm (0.4528)	0.4546 29/64 (0.4531)	0.4578	0.4611	0.4643	0.4675 15/32 (0.4688)
9/16–12	0.4723 12.00 mm (0.4724)	0.4814	0.4868 31/64 (0.4844)	0.4922 12.50 mm (0.4921)	0.4976	0.5030 1/2 (0.5000)	0.5084 13.00 mm (0.5118)
9/16–18	0.5024 1/2 (0.5000)	0.5084	0.5120 13.00 mm (0.5118)	0.5156 33/64 (0.5156)	0.5192	0.5228	0.5264 17/32 (0.5312)

Table 20—Coarse- and Fine-Thread Series[a] —Continued

Screw Size	Theoretical Thread Engagement (%)						
	83 1/3	75	70	65	60	55	50
5/8–11	0.5266	0.5365 13.50 mm (0.5315) 17/32 (0.5312)	0.5424	0.5483 35/64 (0.5469)	0.5542 14.00 mm (0.5512)	0.5601	0.5660 9/16 (0.5625)
5/8–18	0.5649 9/16 (0.5625)	0.5709 14.50 mm (0.5709)	0.5745	0.5787 37/64 (0.5781)	0.5817	0.5853	0.5889 15.00 mm (0.5906)
3/4–10	0.6417 41/64 (0.6406)	0.6526 16.50 mm (0.6496)	0.6591 21/32 (0.6562)	0.6656	0.6721 17.00 mm (0.6693) 43/64 (0.6719)	0.6785	0.6850 17.50 mm (0.6890) 11/16 (0.6875)
3/4–16	0.6823	0.6892 17.50 mm (0.6890) 11/16 (0.6875)	0.6932	0.6973	0.7013	0.7053 45/64 (0.7031)	0.7094 18.00 mm (0.7087)
7/8–9	0.7547 3/4 (0.7500)	0.7668 49/64 (0.7656)	0.7740 19.50 mm (0.7677)	0.7812 25/32 (0.7812)	0.7884 20.00 mm (0.7874)	0.7956 51/64 (0.7969)	0.8028 20.50 mm (0.8071)
7/8–14	0.7977 51/64 (0.7969)	0.8055	0.8101 20.50 mm (0.8071)	0.8147 13/16 (0.8125)	0.8194	0.8240	0.8286 21.00 mm (0.8268) 53/64 (0.8281)
1–8	0.8647 22.00 mm (0.8661) 55/64 (0.8594)	0.8783 7/8 (0.8750)	0.8864 22.50 mm (0.8858)	0.8945 57/64 (0.8906)	0.9026	0.9107 23.00 mm (0.9055) 29/32 (0.9062)	0.9188 59/64 (0.9219)
1–12	0.9098 29/32 (0.9062)	0.9188	0.9242 23.50 mm (0.9252) 59/64 (0.9219)	0.9296	0.9350	0.9404 15/16 (0.9375)	0.9459 24.00 mm (0.9449)
1 1/8–7	0.9704 31/32 (0.9688)	0.9859 63/64 (0.9844)	0.9951	1.0044 1.000	1.0137 1 1/64 (1.0156)	1.0229 26.00 mm (1.0236)	1.0322 1 1/32 (1.0312)
1 1/8–12	1.0348 1 1/32 (1.0312)	1.0439 26.50 mm (1.0433)	1.0493 1 3/64 (1.0469)	1.0547	1.0601	1.0655 27.00 mm (1.0630) 1 1/16 (1.0625)	1.0709
1 1/4–7	1.0954 1 3/32 (1.0938)	1.1109 1 7/64 (1.1094)	1.1201 28.50 mm (1.1220)	1.1294 1 1/8 (1.1250)	1.1387	1.1479 1 9/64 (1.1406)	1.1572 1 5/32 (1.1562)
1 1/4–12	1.1598 29.50 mm (1.1614) 1 5/32 (1.1562)	1.1689	1.1743 1 11/64 (1.1719)	1.1797	1.1851 30.00 mm (1.1811)	1.1905 1 3/16 (1.1875)	1.1959 30.50 mm (1.2008)

[a] See text for instructions for use.

SPECIAL METALS

www.specialmetals.com

U.S.A.

Special Metals Corporation

Billet, rod & bar, flat & tubular products
3200 Riverside Drive
Huntington, WV 25705-1771
Phone +1 (304) 526-5100
+1 (800) 334-4626
Fax +1 (304) 526-5643

Billet & bar products
4317 Middle Settlement Road
New Hartford, NY 13413 5392
Phone +1 (315) 798-2900
+1 (800) 334-8351
Fax +1 (315) 798-2016

Atomized powder products
100 Industry Lane
Princeton, KY 42445
Phone +1 (270) 365-9551
Fax +1 (270) 365-5910

United Kingdom

Special Metals Wiggin Ltd.
Holmer Road
Hereford HR4 9SL
Phone +44 (0)1432 382200
Fax +44 (0)1432 264030

France

Special Metals Services SA
17 rue des Frères Lumière
69680 Chassieu (Lyon)
Phone +33 (0)4 72 47 46 46
Fax +33 (0)4 72 47 46 59

Germany

Special Metals Deutschland Ltd.
Postfach 20 04 09
40102 Düsseldorf
Phone +49 (0)211 38 63 40
Fax +49 (0)211 37 98 64

Hong Kong

Special Metals Pacific Pte. Ltd.
Room 1110, 11th Floor
Tsuen Wan Industrial Centre
220-248 Texaco Road, Tsuen Wan
Phone +852 2439 9336
Fax +852 2530 4511

India

Special Metals Services Ltd.
520, 46th Cross, V Block
Jayanagar, Bangalore 560 041
Phone +91 (0)80 664 6521
Fax +91 (0)80 664 2773

Italy

Special Metals Services SpA
Via Assunta 59
20054 Nova Milanese (MI)
Phone +390 (0)362 4941
Fax +390 (0)362 494224

The Netherlands

Special Metals Services BV
Postbus 8681
3009 AR Rotterdam
Phone +31 (0)10 451 44 55
Fax +31 (0)10 450 05 39

Singapore

Special Metals Pacific Pte. Ltd.
50 Robinson Road
06-00 MNB Building, Singapore 06888
Phone +65 222 3988
Fax +65 221 4298

Affiliated Companies

Special Metals Welding Products
1401 Burris Road
Newton, NC 28658, U.S.A.
Phone +1 (828) 465 0352
+1 (800) 624 3411
Fax +1 (828) 464 8993

Regal Road
Stratford-upon-Avon
Warwickshire CV37 0AZ, U.K.
Phone +44 (0)1789 268017
Fax +44 (0)1789 269681

Controlled Products Group
590 Seaman Street, Stoney Creek
Ontario L8E 4H1, Canada
Phone +1 (905) 643 6555
Fax +1(905) 643 6614

A-1 Wire Tech, Inc.
A Special Metals Company
840 39th Avenue
Rockford, IL 61109, U.S.A.
Phone +1 (815) 226 0477
+1 (800) 426 6380
Fax +1 (815) 226 0537

Reseal SA
A Special Metals Company
200 rue de la Couronne des Prés
78681 Epône Cédex, France
Phone +33 (0)1 30 90 04 00
Fax +33 (0) 30 90 02 11

DAIDO-*SPECIAL METALS* Ltd.
A Joint Venture Company
Daido Building
7-13, Nishi-shinbashi 1-chome
Minato-ku, Tokyo 105, Japan
Phone +81 (0)3 3504 0921
Fax +81 (0)3 3504 0939

TÜV CERT

UKAS

NADCAP

A2LA ACCREDITED