ASM Handbook, Volume 16: Machining
ASM Handbook Committee, p 761-804
DOI: 10.1361/asmhba0002184

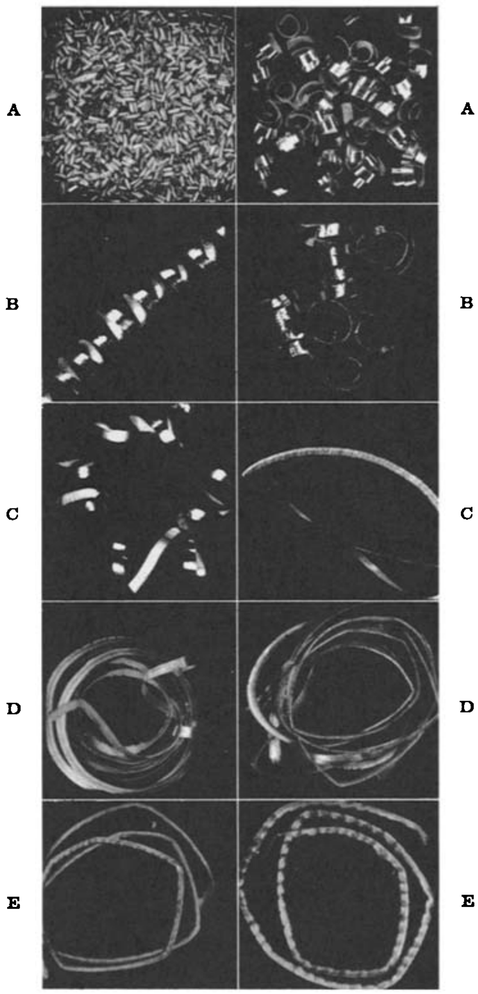
Machining of Aluminum and Aluminum Alloys

ALUMINUM ALLOYS can be machined rapidly and economically. Because of their complex metallurgical structure, their machining characteristics are superior to those of pure aluminum.

The microconstituents present in aluminum alloys have important effects on machining characteristics. Nonabrasive constituents have a beneficial effect, and insoluble abrasive constituents exert a detrimental effect on tool life and surface quality. Constituents that are insoluble but soft and nonabrasive are beneficial because they assist in chip breakage; such constituents are purposely added in formulating high-strength free-cutting alloys for processing in high-speed automatic bar and chucking machines.

In general, the softer alloys—and, to a lesser extent, some of the harder alloys—are likely to form a built-up edge on the cutting lip of the tool. This edge consists of aluminum particles that have become welded to the tool edge because they were melted by the heat generated in cutting. Edge buildup can be minimized by using effective cutting fluids and by employing tools with surfaces that are free of grinding marks and scratches.

Alloys containing more than 10% Si are the most difficult to machine because hard particles of free silicon cause rapid tool wear. Alloys containing more than 5% Si will not finish to the bright machined surfaces of other high-strength aluminum alloys, but will have slightly gray surfaces with little luster. Chips are torn rather than sheared from the work, and special precautions (such as the use of lubricant-containing cutting fluids) must be taken to avoid the buildup of burrs on cutting edges.



Machinability rating	Alloy	Speed, m/min (sfm)	Feed, mm/rev (in./rev) Left photo	Right photo
A	2011-T3	120 (400)	0.066 (0.0026)	0.152 (0.0060)
B	2024-T4	30 (100)	0.152 (0.0060)	0.264 (0.0104)
C	6061-T6	120 (400)	0.152 (0.0060)	0.264 (0.0104)
D	3004-H32	120 (400)	0.152 (0.0060)	0.264 (0.0104)
E	1100-H12	120 (400)	0.152 (0.0060)	0.264 (0.0104)

Fig. 1 Typical chips for machinability ratings A to E (Table 1) for aluminum alloys. All chips were made with 20° rake tool and 2.54 mm (0.100 in.) depth of cut.

Classification of Aluminum Alloys

Cast, wrought, strain hardenable, and heat treatable are the four major classifications of aluminum alloys. Machinability groupings are also used.

Cast alloys containing copper, magnesium, or zinc as the principal alloying elements impose few machining problems. Tools with small rake angles can normally be used with little danger of burring the part or of developing buildup on the cutting edges of tools. Alloys having silicon as the major alloying element require tools with larger rake angles, and they are more economically machined at lower speeds and feeds.

Wrought Alloys. Most wrought aluminum alloys have excellent machining characteristics; several are well suited to multiple-operation machining. A thorough understanding of tool designs and machining practices is essential for full utilization of the free-machining qualities of aluminum alloys.

Strain-hardenable alloys (including commercially pure aluminum) contain no alloying elements that would render them hardenable by solution heat treatment and precipitation, but they can be strengthened to some extent by cold work. In machining, a continuous chip is formed that must be directed away from the workpiece by tools with generous side and back rake angles, thus preventing scratching of the finished surface with the work-hardened chips. These alloys machine easily, although tool pressures are high as a result of high friction. To obtain good surface finish, sharp tools are mandatory because the alloys are gummy. Machinability is improved by cold working; alloys in the full-hard temper are easier to machine to a good finish than those in the annealed condition.

Heat-Treatable Alloys. Most of the alloys of this group contain fairly high percentages of alloying elements such as copper, silicon, magnesium, and zinc. They can be machined to a good finish with or without cutting fluid, but a cutting fluid is recommended for most operations. Turnings usually occur as long, continuous curls, except for the free-machining alloys, which contain chip-breaking constituents. Heat-treatable alloys are more machinable in the heat-treated tempers than in the softer as-fabricated or annealed solution.

Machinability groupings for aluminum alloys are useful in specifying tool forms. For this purpose, alloys are classified into five groups: A, B, C, D, and E, in increasing order of chip length and in decreasing order

Table 1(a) Machinability ratings of wrought aluminum alloys

Alloy designation	Temper	Product form	Hardness, HB (500 kg load, 10 mm ball)	Machinability rating(a)
1060	O	Extruded rod, bar,	19	E
	H12	extruded and drawn	23	E
	H14	tube, pipe	26	D
	H16		30	D
	H18		35	D
1100	O	Sheet, plate; rolled and	23	E
	H12	extruded rod, bar;	28	E
	H14	extruded and drawn	32	D
	H16	tube, pipe; other	38	D
	H18		44	D
2011	T3	Rod, bar, tube, pipe	95	A
	T8		100	A
2014	O	Plate, rod, bar, tube, pipe;	45	D
	T4	other	105	B
	T6		135	B
2017	O	Rolled rod, bar; other	45	C
	T4		105	B
2018	T61	Forging stock	120	B
2024	O	Sheet, plate, rod, bar, tube,	47	D
	T3	pipe; other	120	B
	T4		120	B
	T61		130	B
2025	T6	Forging stock	110	B
2117	T4	Rivet wire, rod	70	C
2218	T72	Forging stock	95	B
2219	O	Sheet, plate; extruded rod,	...	...
	T42	bar; extruded and drawn	...	...
	T351	tube, pipe; forging stock	100	B
	T37		117	B
	T62		115	B
	T851		130	B
	T87		130	B
2618	T61	Forgings	115	B
3002	O	Sheet	25	...
	H25		...	...
3003	O	Sheet, plate; rolled and	28	E
	H12	extruded rod, bar;	35	E
	H14	extruded and drawn	40	D
	H16	tube, pipe; other	47	D
	H18		55	D
3004	O	Sheet, plate; drawn tube,	45	D
	H32	pipe	52	D
	H34		63	C
	H36		70	C
	H38		77	C
4032	T6	Forging stock	120	B
5005	O	Sheet, plate; rolled rod and	28	E
	H12	bar; other	36	E
	H14		41	D
	H16		46	D
	H18		51	D
	H32		36	E
	H34		41	D
	H36		46	D
	H38		51	D
5050	O	Sheet, plate; drawn tube,	36	E
	H32	pipe	46	D
	H34		53	D
	H36		58	C
	H38		63	C
5052	O	Sheet, plate; rolled rod,	47	D
	H32	bar; drawn tube, pipe;	60	D
	H34	other	68	C
	H36		73	C
	H38		77	C
5056	O	Rivet rod, wire	65	D
	H18		105	C
	H38		100	C
5083	O	Sheet, plate, rod, bar, tube,	67	D
	H321	pipe, forgings	82	D
5086	O	Sheet, plate; extruded rod,	60	D
	H32	bar; extruded and drawn	72	D
	H34	tube, pipe	82	C
	H112		64	D
5154	O	Sheet, plate; welding wire	58	D
	H32	and rod	67	D
	H34		73	C
	H36		78	C
	H38		80	C
	H112		63	D
5252	H25	Sheet	68	C
	H38		75	C
5254	O	Sheet, plate	58	D
	H32		67	D
	H34		73	C
	H36		78	C
	H38		80	C
	H112		63	D
5257	H25	Sheet	32	C
	H28		43	C
5357	O	Sheet, plate	32	D
	H25		50	C
	H28		55	C
5454	O	Sheet, plate; other	62	D
	H32		73	D
	H34		81	C
	H111		70	D
	H112		62	D
5456	O	Sheet, plate; extruded rod	70	D
	H111	bar; extruded tube, pipe;	75	D
	H112	forgings	70	D
	H116		90	D
5457	O	Sheet	32	E
	H25		48	C
	H28		55	C
5557	O	Sheet	27	E
	H25		46	D
	H28		55	D
5652	O	Sheet, plate	47	D
	H32		60	D
	H34		68	C
	H36		73	C
	H38		77	C
5657	H25	Sheet	40	D
	H28		50	D
6005	T5	Extruded rod, bar	95	C
6061	O	Sheet, plate, rod, bar, tube,	30	D
	T4	pipe; forging; other	65	C
	T6		95	C
6063	O	Extruded rod, bar;	25	D
	T1	extruded and drawn	42	D
	T4	tube, pipe	60	D
	T5		60	C
	T6		73	C
	T83		82	C
	T831		70	C
	T832		95	C
6066	O	Extruded rod, bar; forging	43	D
	T4	stock	90	C
	T6		120	B
6070	T6	Rod, bar, tube, pipe	120	C
6151	T6	Forging stock	100	...
6262	T9	Rod, bar, tube, pipe	120	B
6463	T1	Extruded rod, bar;	42	D
	T5	extruded and drawn	60	C
	T6	tube, pipe	74	C
6951	O	Sheet	28	...
	T6		82	...
7001	O	Extruded rod, bar	60	B
	T6		160	B
7005	T53	Rod, bar, tube, pipe	...	...
7075	O	Sheet, plate, rod, bar, tube,	60	D
	T6	pipe; forging stock	150	B
7079	O	Sheet, plate, rod, bar, tube,	...	...
	T6	pipe; forging stock	145	B
7178	O	Sheet, plate, rod, bar, tube,	60	...
	T6	pipe	160	B
	T76		...	...
8280	O	Sheet, plate	...	B
	H12		...	A

(a) A, B, C, D, and E are relative ratings in increasing order of chip length (see Fig. 1) and decreasing order of quality of finish. A, free cutting, very small broken chips and excellent finish; B, curled or easily broken chips and good-to-excellent finish; C, continuous chips and good finish; D, continuous chips and satisfactory finish; E, optimum tool design and machine settings required to obtain satisfactory control of chip and finish. Source: Ref 1

Table 1(b) Machinability ratings of cast aluminum alloys

Alloy designation	Temper	Casting type	Hardness, HB (500 kg load, 10 mm ball)	Machinability rating(a)	Alloy designation	Temper	Casting type	Hardness, HB (500 kg load, 10 mm ball)	Machinability rating(a)
208	F	Sand	55	B	A356	T6	Permanent mold	80	. . .
213	F	Permanent mold	85	. . .		T61	Permanent mold	80	B
222	T52	Permanent mold	100	. . .	357	F	Permanent mold	. . .	. . .
	T551	Permanent mold	115	. . .		T51	Permanent mold	. . .	. . .
	T65	Permanent mold	140	. . .		T6	Sand	90	B
238	F	Permanent mold	100	B		T6	Permanent mold	85	B
A240	F	Sand	90	A		T7	Sand	60	. . .
242	F	Sand	. . .	. . .		T7	Permanent mold	70	. . .
	T21	Sand	70	B	A357	T6	Sand	85	. . .
	T571	Sand	85	B		T6	Permanent mold	85	. . .
	T571	Permanent mold	105	B	B358	T6	Permanent mold	90	B
	T61	Permanent mold	110	B		T62	Permanent mold	. . .	. . .
	T77	Sand	75	B	359	T61	Permanent mold	90	. . .
A242	T77	Sand	70	. . .		T62	Permanent mold	100	B
295	T4	Sand	60	B	360	F	Die	75	C
	T6	Sand	75	B	A360	F	Die	75	C
	T62	Sand	90	B	364	F	Die	. . .	C
B295	T4	Permanent mold	75	B	380	F	Die	80	B
	T6	Permanent mold	90	B	A380	F	Die	80	B
	T7	Permanent mold	80	B	384	F	Die	. . .	C
308	F	Permanent mold	70	B	390	F	Die	120	. . .
319	F	Sand, permanent mold	70	C	A390	F	Sand	100	. . .
	T5	Sand	80	B		F	Permanent mold	110	. . .
	T6	Sand	80	B		T5	Sand	100	. . .
	T6	Permanent mold	95	B		T5	Permanent mold	110	. . .
A332	T551	Permanent mold	105	C		T6	Sand	140	. . .
	T65	Permanent mold	125	C		T6	Permanent mold	145	. . .
F332	T5	Permanent mold	105	C		T7	Sand	115	. . .
333	F	Permanent mold	90	C		T7	Permanent mold	120	. . .
	T5	Permanent mold	100	B	413	F	Die	80	E
	T6	Permanent mold	105	B	A413	F	Die	80	. . .
	T7	Permanent mold	90	B	443	F	Sand	40	E
354	T61	Permanent mold	100	B		F	Permanent mold	45	E
	T62	Permanent mold	110	B		F	Die	50	E
355	F	Sand	. . .	. . .	A444	F	Sand	. . .	. . .
	T51	Sand	65	B		F	Permanent mold	44	. . .
	T51	Permanent mold	75	B		T4	Sand	. . .	. . .
	T6	Sand	80	B		T4	Permanent mold	45	. . .
	T6	Permanent mold	90	B	514	F	Sand	50	B
	T61	Sand	90	B	A514	F	Permanent mold	60	B
	T62	Permanent mold	105	B		F	Die	. . .	B
	T7	Sand	85	B	B514	F	Sand	50	. . .
	T7	Permanent mold	85	B	F514	F	Sand	50	B
	T71	Sand	75	B	L514	F	Die	. . .	. . .
	T71	Permanent mold	85	B	518	F	Die	80	B
C355	T6	Sand	85	. . .	520	T4	Sand	75	B
	T6	Permanent mold	90	. . .	535	F	Sand	70	. . .
	T61	Permanent mold	100	B	A535	F	Sand	65	B
356	F	Sand	. . .	. . .	B535	F	Sand	65	A
	F	Permanent mold	. . .	. . .	705	F	Sand	65	B
	T51	Sand	60	C	707	F	Sand	85	B
	T51	Permanent mold	. . .	C	A712	F	Sand	75	B
	T6	Sand	70	C	C712	F	Permanent mold	70	B
	T6	Permanent mold	90	C	D712	F	Sand	75	B
	T7	Sand	75	C	713	F	Sand	75	B
	T7	Permanent mold	70	C	850	T5	Sand	45	A
	T71	Sand	60	C		T5	Permanent mold	45	A
A356	F	Sand	. . .	. . .	A850	T5	Sand, permanent mold	45	A
	T51	Sand	. . .	. . .	B850	T5	Sand	65	A
	T6	Sand	75	. . .		T5	Permanent mold	70	A

(a) A, B, C, D, and E are relative ratings in increasing order of chip length (see Fig. 1) and decreasing order of quality of finish. A, free cutting, very small broken chips and excellent finish; B, curled or easily broken chips and good-to-excellent finish; C, continuous chips and good finish; D, continuous chips and satisfactory finish; E, optimum tool design and machine settings required to obtain satisfactory control of chip and finish. Source: Ref 1

of finish quality, as defined in the footnotes of Tables 1(a) and 1(b). Ratings for most commercial aluminum alloys are given in Tables 1(a) and 1(b), and typical chips for each rating are illustrated in Fig. 1.

Cutting Force and Power

The cutting force, and therefore the power, required to machine aluminum is less than might be expected on the basis of its mechanical properties. Although the cutting force required to machine similar metals is often in direct proportion to tensile strength, this proportion is not necessarily valid with dissimilar metals. For example, the common mechanical properties of 2017-T4 aluminum alloy and of hot-rolled low-carbon steel are quite similar (Table 2), but as Fig. 2 shows, the cutting force required in turning aluminum is only about 35% of that required in turning low-carbon steel. Consequently, as shown in Fig. 3, the number of cubic millimeters of metal that can be removed per minute per unit kilowatt expended is approximately three times as great for aluminum alloy 2017-T4 as for hot-rolled low-carbon steel of closely similar tensile strength.

Table 2 Comparison of common mechanical properties of 2017-T4 aluminum alloy and hot-rolled low-carbon steel

Mechanical property	Aluminum alloy 2017-T4	Hot-rolled low-carbon steel
Tensile strength, MPa (ksi)	440 (63.8)	449 (65.2)
0.2% yield strength, MPa (ksi)	278 (40.3)	277 (40.2)(a)
Elongation in four diameters, %	26.0	32.7
Shear strength, MPa (ksi)	279 (40.5)	324 (47.0)
Hardness, HB		
500 kg, 10 mm ball	113	110
3000 kg, 10 mm ball	132	128

(a) Yield point is 309 MPa (44.8 ksi).

Selection of Alloy and Temper

An application often dictates the use of a specific alloy or temper or both. Under these conditions, composition cannot be changed for the sake of improving machinability. However, there is often a marked difference in machinability among different tempers of the same alloy. Therefore, it may be feasible to do some or all of the machining operations with the alloy in the most favorable condition for machining and then to convert the alloy to the temper specified for the end use.

For some applications, two or more alloys are equally acceptable. Under these conditions, machinability can be a major consideration in making the final selection. For example, high-strength, free-cutting alloy 2011 can be machined to an excellent surface finish at high speed and feed, with a low rate of tool wear. The chips formed are finely broken. Alloy 2011 is therefore recommended for all general and high-production machining where a free-cutting alloy is desired. Alloy 2011 is especially desirable for multiple-operation machining, mainly because it machines with a broken chip. Stock for multiple-operation machining is also available in alloys 2017, 2024, 6061, and 6262 in several heat-treated tempers.

Alloys 2024-T4, 2017-T4, and 7075-T6 produce continuous chips that must be broken by a chip breaker in the tool. Alloys 6061-T6 and 5056-H38 are slightly more difficult to machine, and they produce chips that are difficult to control. The softer alloys 5052, 3003, and 1100 are likely to produce gummy chips. Wrought alloy 4032 and cast alloys 220, 13, and A132 are quite abrasive, and high rates of tool wear result. The following example describes an application in which a change to an alloy and temper having better machinability improved results.

Example 1: Change of Alloy 6061-T6 to 6262-T9 to Improve Machinability of Hexagonal Nut. When alloy 6061-T6 was used for a hexagonal nut (Fig. 4), a lead of three threads chamfer on the tap form was required to produce a thread without tearing. A length of 4.8 mm (3/16 in.) had to be cut off after tapping. After changing to the freer-cutting 6262-T9, lead was reduced to 1½ threads, so that only 3.2 mm (1/8 in.) had to be cut off. Material savings was 8.5%. Because 6262-T9 produced fewer burrs on the tapped threads, frequency of tool grinding was reduced.

With 6061-T6, the workpiece before tapping weighed 0.0254 kg (0.0560 lb), and with 6262-T9, it weighed 0.0232 kg (0.0512 lb). Production rate was 890 pieces per hour with both alloys.

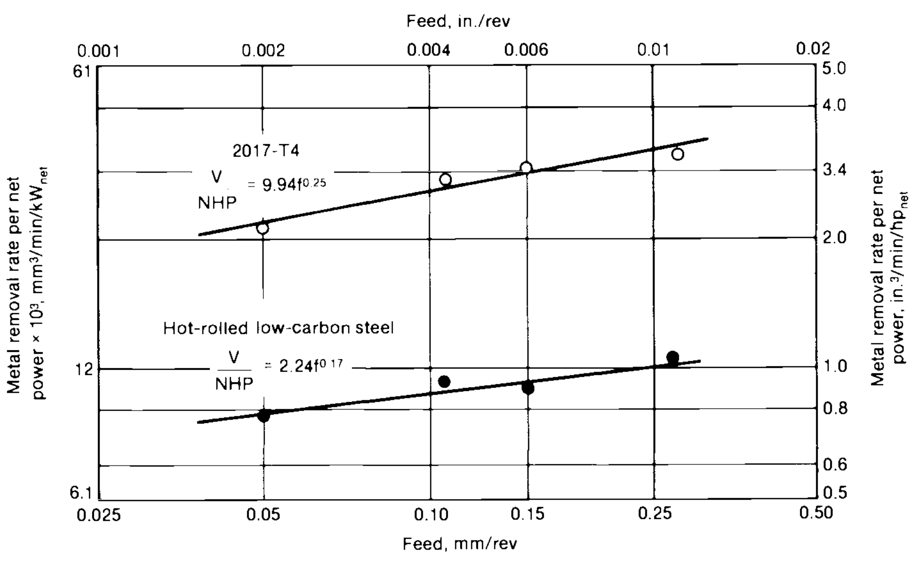


Fig. 3 The effect of feed on metal removal rate per net power for low-carbon steel and aluminum alloy 2017-T4 with comparable mechanical properties. V/NHP, volume per net horsepower

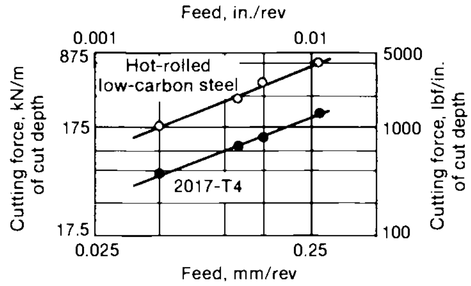


Fig. 2 The effect of feed on cutting force for low-carbon steel and aluminum alloy 2017-T4 with comparable mechanical properties

Bar Versus Tubing

Many parts can be machined equally well from either bar stock or tubing. When either of these product forms can be used, cost per piece machined is the determining factor. Initial cost of bar versus tube, cost of additional machining to make the part from bar stock, and value of the additional scrap that results from machining bar stock must be considered to determine cost. Because of the high cost of producing small tubing, total cost per piece machined may be greater when tubing of small sizes (less than about 32 mm, or 1¼ in., in diameter) is used. For parts requiring a diameter greater than 32 mm (1¼ in.), tubing is usually less expensive.

General Machining Conditions

Power requirements for machining are proportional to speed and cutting force, and the power lost in the bearings and gears of the machine increases with speed. Power requirements for machining aluminum decrease somewhat as the rake angle of the cutting tool is increased. Table 3 lists typical power requirements for several wrought and cast alloys, as measured at the cutter with single-point tools having 0 and 20° rake angles.

The cutting force for aluminum alloys can vary widely at low speeds, such as 30 to 60 m/min (100 to 200 sfm), rising momentarily to peak values several times higher than normal. At higher speeds, the cutting

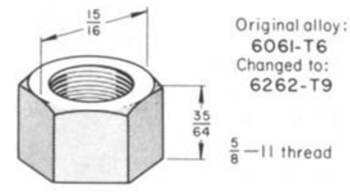


Fig. 4 Hexagonal nut for which a change from alloy 6061-T6 to 6262-T9 resulted in a savings of material and less burring of the tapped thread. Dimensions given in inches

Table 3 Power requirements for machining aluminum alloys

Alloy	Power: 0° rake (kW/mm³/min) × 10⁻⁴	Power: 0° rake hp/in.³/min	Power: 20° rake (kW/mm³/min) × 10⁻⁴	Power: 20° rake hp/in.³/min
F132-T5	0.16–0.32	0.20–0.40	0.16–0.41	0.20–0.50
356-T51	0.20–0.36	0.25–0.45	0.24–0.81	0.30–1.00
2011-T3	0.12–0.20	0.15–0.25	0.16–0.24	0.20–0.30
2024-T351	0.16–0.32	0.20–0.40	0.24–0.41	0.30–0.50
6061-T651	0.20–0.28	0.25–0.35	0.24–0.41	0.30–0.50

Table 4 Design of single-point tools for machining aluminum alloys of A and B machinability ratings

Tool details	High-speed steel: Roughing	High-speed steel: Finishing	Carbide: Roughing	Carbide: Finishing
Back rake, degrees	20	20	20	20
Side rake, degrees	20	20	20	20
End relief, degrees	10	10	7	7
Side relief, degrees	10	10	7	7
End cutting edge, degrees	5	5	5	5
Side cutting edge, degrees	10	10	10	10
Nose radius, mm (in.)	1.6 (0.063)	5.1 (0.20)	1.6 (0.063)	5.1 (0.20)

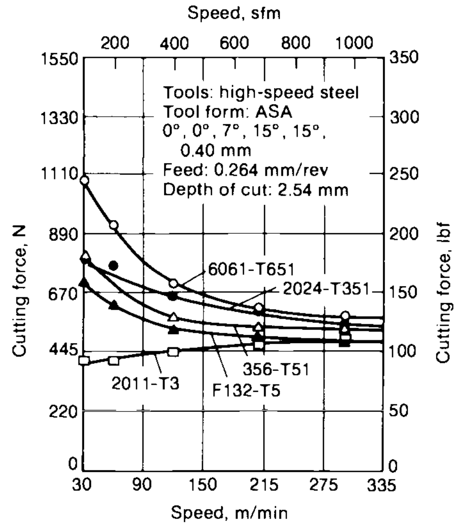


Fig. 5 The effect of speed on cutting force for five aluminum alloys

force for machining 2011-T3 alloy rises slightly with increasing speed, but for most alloys it decreases. The overall effect of speed on cutting force is small. As speed increases up to about 300 m/min (1000 sfm), cutting force changes slightly; above 300 m/min (1000 sfm), the effect of speed is negligible. Increasing the speed does not produce much more heat, but it does shorten the time available for removing the heat from the tool. The effect of speed on cutting force for several aluminum alloys is plotted in Fig. 5. Heating of tool surfaces is not sufficient to have a harmful effect on a high-speed steel tool until the speed exceeds about 215 m/min (700 sfm). High-speed steel can be used for speeds well beyond this limit, but carbide tools are recommended for long tool life.

The cutting speed for aluminum alloys is determined by the limits of the machine tool and by the workpiece. Speeds as high as 4600 m/min (15 000 sfm) have been used in aerospace applications (see the article "High-Speed Machining" in this Volume). Even higher speeds have been achieved with experimental equipment. However, in most practice, mainly because of the limitations imposed by available spindle speed, available horsepower, and dynamic balance of the part, machining speeds are seldom higher than 900 m/min (3000 sfm) and they are more commonly less than 300 m/min (1000 sfm), as indicated by the examples in this article.

Cutting speed should be as high as is practical in order to save time and to minimize temperature rise in the part, as described in the section "Thermal Expansion" in this article. As cutting speed is increased above 30 to 60 m/min (100 to 200 sfm), the probability of forming a built-up edge on the cutter is reduced, chips break more readily, and finish is improved.

Depth of cut should be as great as possible within the limits of part strength, chucking equipment, power of the machine tool, and amount of stock to be removed in order to minimize the number of cuts required. As depth of cut is increased, cutting force increases. Depth of cut must be limited to a value that will not distort the workpiece or cause it to slip, nor overload the machine. Depth of cut in roughing may be as high as 6.35 mm (0.250 in.) for small work or up to 38.10 mm (1.500 in.) for medium or large work. At the opposite extreme, depth of cut in finishing is often less than 0.635 mm (0.025 in.).

Feed will depend on the finish desired and on the strength and rigidity of the workpiece and of the machine. Finishing cuts require a light feed of 0.05 to 0.15 mm/rev (0.002 to 0.006 in./rev); rough cuts may use a feed of 0.15 to 2.03 mm/rev (0.006 to 0.080 in./rev). Alloys with machinability ratings of D and E are best machined with a feed in the lower end of the range.

Table 5 Current machining practice for 390 aluminum alloy

Operation	Tool	Speed, mm/min (sfm)	Feed	Depth, mm (in.)	Coolant
Turning	Tungsten carbide, with J polish (SPG 422 insert)	30–150 (100–500)	0.13–0.51 mm/rev (0.005–0.02 in./rev)	0.13–5.1 (0.005–0.20)	Soluble oil (20:1)
	Diamond (SPG 422 insert)	310–910 (1000–3000)	0.08–0.38 mm/rev (0.003–0.015 in./rev)	0.13–3.8 (0.005–0.15)	Soluble oil (20:1)
Milling	Tungsten carbide, C-2 or C-3, 5–10° rake, 7–15° shear	99–175 (325–575)	0.10–0.30 mm/tooth (0.004–0.012 in./tooth)	0.13–5.1 (0.005–0.20)	Soluble oil (15:1) or dry
	Diamond, 5–10° rake, 7–15° shear	400–790 (1300–2600)	0.10–0.20 mm/tooth (0.004–0.008 in./tooth)	0.20–1.0 (0.008–0.040)	Soluble oil (15:1) or dry
Drilling	High-speed steel, high helix	20 (70)	381 mm/min (15 in./min)	6 × diam	Soluble oil (20:1), 19 L/min (5 gal./min) at 170 kPa (25 psi)
	Tungsten carbide tipped, coolant fed	90 (300)	508 mm/min (20 in./min)	6 × diam	Soluble oil (20:1), 34 L/min (9 gal./min) at 3.4 MPa (500 psi)
Tapping	Form taps, with four lobes	20 (70)	By pitch	6 × diam	Soluble oil (20:1), 19 L/min (5 gal./min) at 170 kPa (25 psi)
Broaching	Tungsten carbide, 0° shear, 5° rake	45 (150)	229 mm/min (9 in./min)	0.13 (0.005)	Soluble oil (20:1) or dry

Source: Ref 2

Table 6 Cutting fluids for aluminum

Type of lubricant	Principal ingredients	Viscosity range	Application; maintenance	Relative effectiveness	Necessary precautions
Mineral oils (fatty-additive type preferred)	Mineral oil, lard, or neats-foot oil; oleic acid or butyl stearate	40 SUS at 40 °C (100 °F) for high-speed machining to 300 SUS for low speeds	Generous flow at all cutting edges; keep recirculating fluid clean and cool.	Good lubricity and chip flushing; fair cooling; excellent finish as built-up edge is minimized	Control air above oil where mist application endangers shop air; remove oil from finished parts (also from chips to reclaim and reduce fire hazard).
Soluble oils (emulsions) (3–5% soluble oil in water)	Soluble oil, petroleum sulfonate emulsifying agents, water (oil is added); rust inhibitor, germicide, stain inhibitor	Generally low	Generous flow at all cutting edges; keep recirculating fluid clean; cool when necessary.	Good chip flushing; lubrication adjustable by varying concentration; excellent cooling; good finish	For high-speed machining, cooling is more important than lubrication; where emulsion is applied as mist, keep oil content as low as possible to reduce air and shop contamination.
Aqueous chemical solutions	Water; soluble synthetics (usually clear); sometimes, fatty materials; rust inhibitors; germicides	Generally low	Generous flow at all cutting edges; keep recirculating fluid clean; cool as required.	Good chip flushing; excellent visibility of cut; excellent cooling; adjustable lubrication; good finish	Keep oil content low; control mist; consider cost (significantly higher than soluble-oil emulsions).
Stick lubricants	Mineral compounds, animal fats, waxes, synthetics	Various hardnesses	Applied as required to blades, wheels, disks, or files or to workpiece	Prevents the loading of abrasive surfaces or of teeth of saws and files	Application is intermittent, but should be monitored and made as required throughout run.

Source: Ref 1

Tool Design

Tools intended for machining aluminum and its softer alloys should be ground to allow considerably more side rake and back rake than are customary when machining steel. Therefore, they approach the contours of tools designed for cutting hardwood. The larger rake angles are recommended for finishing tools and for the machining of alloys that are not free cutting, especially the softer alloys, which require exceptionally acute and keen cutting edges. Smaller rake angles can be used for the free-cutting alloys and for roughing cuts, which require a sturdy tool for the heavier cuts and feeds employed. Suggested rake angles, as related to alloy machinability rating (Table 1), are:

Rating	Rake angle, degrees
A	0–20
B	20
C	20–30
D	40
E	40

Tool forms for machining aluminum alloys with machinability ratings of A and B with single-point tools are given in Table 4. Variations of these may be desirable, depending on machining conditions or shape of the workpiece.

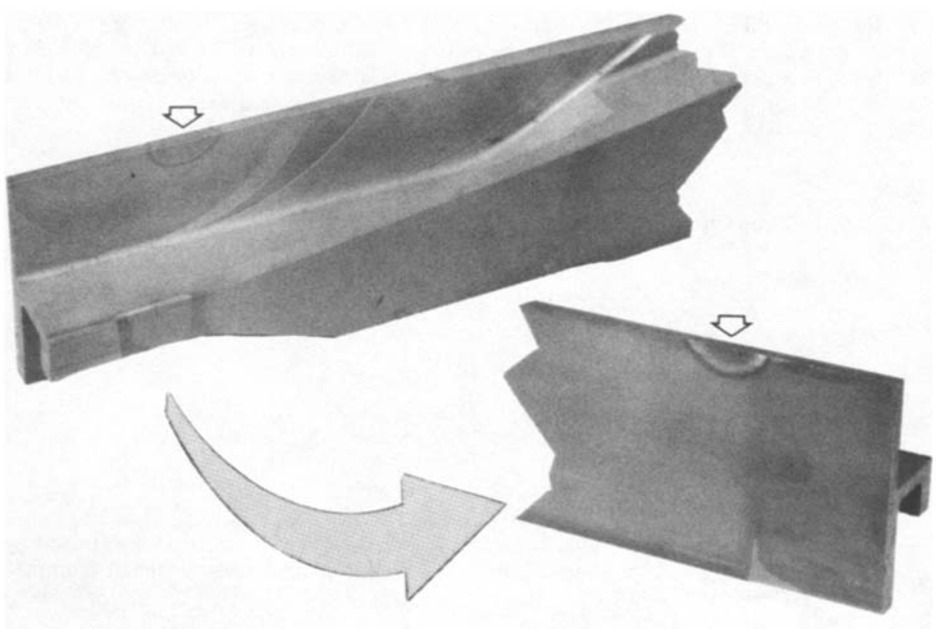


Fig. 6 Typical spar mill defect at a section change. Courtesy of McDonnell Douglas Canada, Ltd.

Clearance angle is important to proper functioning of the tool. Too small a clearance will permit the side or heel of the tool to rub the work and generate heat, while too large an angle will cause the tool to dig into the work and chatter. This angle must be carried around the side of the tool that advances into the work. For most applications, clearance angles of 6 to 10° are suitable. The side rake angle imparts to the tool a slicing action that assists materially in shearing the chip from the stock. This angle is important and should be held within the ranges recommended in subsequent sections of this article.

Cutting Edge Finish. For maximum performance, it is essential that tool cutting edges be keen, smooth, and free of grinding-wheel scratches, burrs, or wire edges. Keen edges can be obtained by finish grinding on a fine abrasive wheel and then lapping, or hand stoning with a fine oilstone. Neither the angles nor the contour of the cutting edge should be appreciably modified during tool finishing.

Tool Material

Water-hardening tool steel, such as W1, heat treated to 65 to 68 HRC, is an adequate cutting tool material when production runs are short and speeds are low. However, tool steels of this type soften rapidly if the temperature of the cutting edge exceeds 150 °C (300 °F). In addition, tools made from water-hardening tool steel have low resistance to edge wear.

High-speed steel tools are generally satisfactory for machining all but the high-silicon alloys, which are quite abrasive and should be machined with carbide or diamond tools, unless runs are short. For short runs, high-speed steel is usually satisfactory. Complexly shaped cutters such as twist drills, reamers and counterbores, taps and other thread-cutting tools, end mills and many other types of milling cutters, and form tools are widely used in turning operations. For general machining and for relatively short-run production, single-point tools of high-speed steel are also commonly used.

Grade M2 high-speed tool steel is commonly used in machining aluminum, although M7 is sometimes preferred because it is more suitable for fine-edge tools and is more abrasion resistant. Grade T2 is also used sometimes, as are high-speed steels containing cobalt.

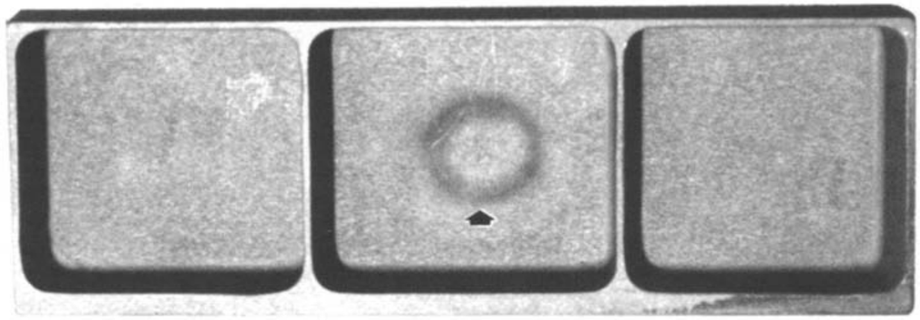
Carbide Tools. Because of the brittleness of the tool tip, the lip angle for carbide tools is usually greater than those recommended for high-speed steel tools in order to provide maximum support to the edge. This is indicated in Table 4, in which smaller relief angles are given for carbide tools. The rake angles can also be decreased to zero or negative to increase the lip angle, but negative rake angles are generally not recommended. When very light finishing cuts are made, it is sometimes feasible to reverse this practice and use higher rake angles and smaller lip angles.

Carbide tools retain sharp edges over a longer period between regrinds than carbon or high-speed steel tools, provided they are not used for heavy, intermittent cuts. A better finish is obtained because of the hardness of the tool tip compared to that of the stock. Carbide tools are particularly useful for machining high-silicon alloys, many of which cannot otherwise be machined satisfactorily under production conditions.

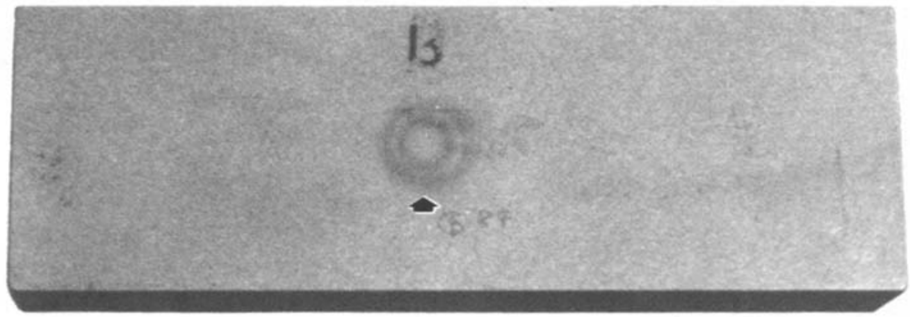
For longer production runs, cemented tungsten carbide tooling is generally the choice for machining low-silicon alloys and is often used for high-silicon alloys. The primary advantage offered by carbides in the machining of easily machined aluminum alloys is the greatly increased cutting speed possible compared to high-speed steel tooling—double or more the metal removal rates possible with the latter.

Both brazed-tip and indexable inserts are suitable. Straight-tungsten carbides are recommended in the C-2 to C-3 application categories. Some carbide specialists recommend fine-grain carbides for aluminum because a keener edge can be produced and because any edge chipping is small and therefore has a smaller effect on sharpness. Edge sharpness is desirable for cutting aluminum.

Coatings, which have proved to be highly advantageous in many steel machining operations, do not appear to provide any benefits in the machining of aluminum alloys. Indeed, vacuum-deposited coatings have a detrimental effect on the smoothness of the rake face of an insert, and all aluminum machining specialists stress the need for highly polished cutters in any material.



(a)



(b)

Fig. 7 Typical end mill defect associated with vertical plunge cut pocketing operations. (a) Front view. (b) Back view. Courtesy of McDonnell Douglas Canada, Ltd.

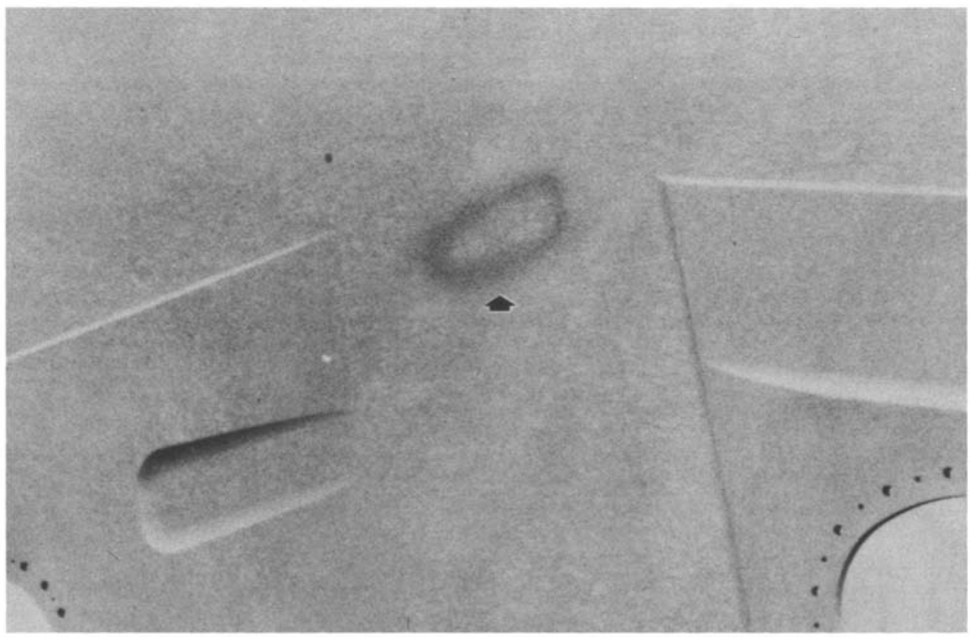


Fig. 8 Milling defect associated with the NC machining of 7075-T7651 aluminum plate with a 75 mm (3 in.) diam milling cutter. Courtesy of McDonnell Douglas Canada, Ltd.

Diamond tools are used only in operations requiring an exceptionally high finish, particularly on high-silicon alloys, in which particles of free silicon will in time slightly dull the cutting edge of even carbide tools. Finishing cuts with diamond tools seldom exceed a few thousandths of an inch.

Single-crystal natural diamond cutting tools have long been used for producing very smooth finishes on turned aluminum workpieces—finishes of the order of 0.125 μm (5 μin.) roughness and better. For example, the lens barrels of even relatively inexpensive cameras are often turned with

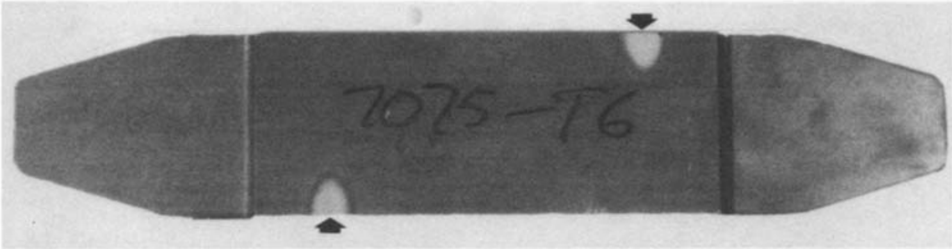


Fig. 9 Attachment tee made of 7075-T6 aluminum with the soft spot area having a hardness of 73 to 83 HRB compared to 90 HRB for the rest of the structure. Courtesy of McDonnell Douglas Canada, Ltd.

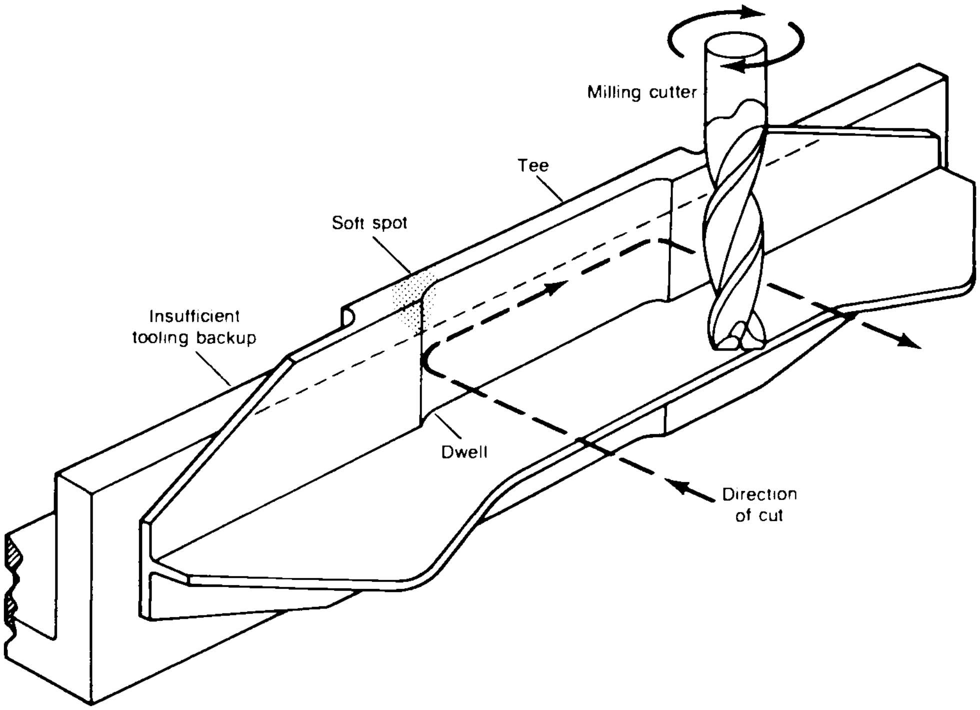


Fig. 10 Schematic of the milling cutter path in machining the attachment tee shown in Fig. 9

single-crystal diamond tools, and no further finishing operations are performed.

The principal advantages of single-crystal diamond for machining aluminum are the ability to take an extremely fine edge, the highest known hardness and wear resistance, and an extremely low coefficient of friction with no tendency to adhere. In addition, there is no chemical reaction between the carbon of diamond and aluminum.

Diamond tools are usually made with either circular or faceted cutting edges, the latter being the more common. With the faceted cutter, there may be as many as five facets on one cutting edge, each varying in size from 0.5 to 1.5 mm (0.02 to 0.06 in.). Cutting angles of 74 to 90° and top rake angles of 6 to 10° are used. Rake should not be less than −6°. The tool should be set on, or slightly above, the centerline of the work.

Polycrystalline Diamond Tooling (Ref 1). A development of recent years that is having a major effect in many aluminum machining operations has been the introduction of compacted polycrystalline diamond cutting tools. Polycrystalline diamond is produced by a technique that includes pressing gritlike diamond particles in a mold under pressures and temperatures approaching those necessary to convert carbon into synthetic diamond. Natural or synthetic diamond particles can be used as the raw material, and it is possible to produce a compacted polycrystalline diamond surface on a tungsten carbide substrate.

Polycrystalline diamond, being true diamond, possesses the basic physical and chemical properties that make single-crystal diamond a valuable tool material for machining aluminum, but it does differ in several respects. Because it is polycrystalline, it cannot produce as fine an edge as single-crystal diamond. Therefore, it is not suitable for producing the finest finishes; about 0.125 to 0.25 μm (5 to 10 μin.) is the practical limit, which is acceptable in most applications.

Because the crystal orientation is random, impact resistance is high, and the properties are essentially omnidirectional. In contrast, single-crystal stones require careful orientation of the crystal for optimum utilization of properties. Another advantage over natural stones is that tool tips can be produced virtually to final shape, saving considerable effort for the toolmaker, and the use of a carbide substrate permits easy brazing of the polycrystalline tip to a tool steel shank or standard-size insert.

Initially introduced in the United States in 1972, polycrystalline diamond tooling has gained an important position in the machining of the highly abrasive silicon-aluminum alloys. Typical production operations include turning, boring, and milling. Reports of polycrystalline diamond tools outlasting tungsten carbide by factors as great as 100 in the machining of high-silicon aluminum are not uncommon. More detailed information on polycrystalline diamond tooling can be found in the article "Ultrahard Tool Materials" in this Volume. Application data for aluminum-silicon alloys follow.

Machining of High-Silicon Aluminum Alloy 390 (Ref 2)

Microstructural Considerations. Because of the presence of primary silicon in the microstructure, the machining of aluminum-silicon alloys such as 390 must be approached somewhat differently from the machining of conventional aluminum casting alloys. The major alloying ingredients are 17% Si, 4.5% Cu, and 0.5% Mg. The eutectic composition is approximately 12% Si; the remaining 5 to 6% Si occurs as a primary phase. This primary-phase silicon is desirable in wear parts, but it affects cutting tool life drastically when machining alloy 390. The 390 silicon-aluminum alloy machines differently from other cast aluminum alloys, such as 319 (5.5 to 6.5% Si), and 380 (7.5 to 9.5% Si), in that the primary silicon causes rapid tool wear. Alloy 390 was originally developed for use in engines in which the cylinder bore is finished in such a manner that primary silicon stands in relief and prevents piston rings, and so on, from contacting the aluminum matrix.

Most aluminum casting alloys contain silicon as a major alloying element. Silicon, in either eutectic or primary form, is much harder than any other phase of the microstructure. The hardness of silicon crystals generally ranges from 1000 to 1300 HK, while the microhardness of an aluminum casting alloy matrix seldom exceeds 180 HK. Silicon, like the heavy-element intermetallic impurity phases of iron, manganese, and chromium, is a very abrasive material in an otherwise soft matrix and is

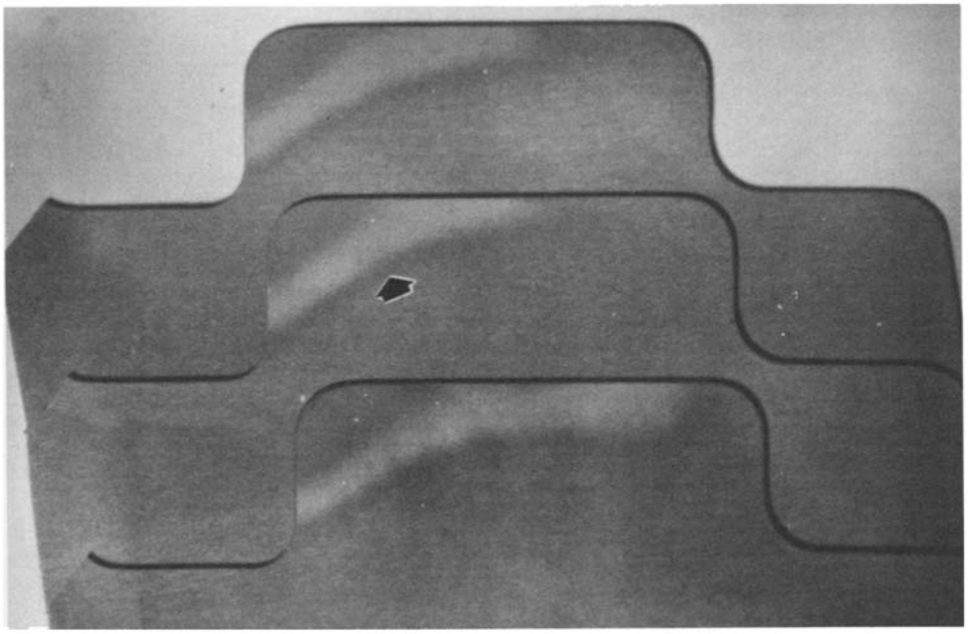


Fig. 11 Occurrence of soft spots in the same location on each of three 7075-T651 aluminum plate components. Of the 19 parts produced, 13 had a soft zone hardness of 68 to 87 HRB and conductivity of 36 to 39%. Unaffected regions of the plate had a hardness of 90 to 91 HRB and conductivity of 31 to 32%. Courtesy of McDonnell Douglas Canada, Ltd.

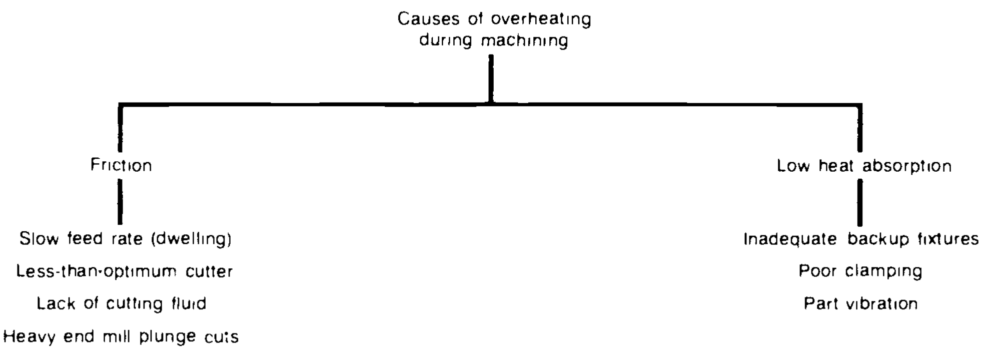


Fig. 13 Causes of overheating during machining process attributable to friction and low heat absorption problems

the element that singularly has the greatest tendency to decrease cutting tool life.

A fine, well-modified eutectic silicon structure is far less detrimental to tool life than heavy-element intermetallic impurity phases. However, the rate of wear on cutting tools increases as silicon particle size increases. If the eutectic silicon structure is coarse, tool life suffers. Primary silicon crystals, even if well refined and distributed, are more detrimental than eutectic silicon, and large unrefined primary crystals can seriously degrade tool life.

In permanent mold or sand casting, primary silicon is controlled by a refinement treatment, adding a small amount of phosphorus to the molten alloy. Unrefined primary silicon is eight to ten times the size of refined silicon crystal, and it drastically affects machinability.

In conventional die casting, a refinement treatment is not necessary. Primary silicon size in conventional die castings is very small even without phosphorus refinement, and the silicon size and distribution are controlled by such process parameters as melt temperature, die temperature, and die fill rate.

Regardless of the casting method employed, primary silicon acts as a chip breaker. Although primary silicon crystals are very hard, they are also quite friable. When the cutting tool passes through the matrix of the alloy, it fractures primary silicon particles. This fracturing, along with the natural hardness of the alloy and work hardening of the chips, causes the close, tightly curled chips to break into short ringlets similar in appearance to gray iron chips.

Machining Parameters. Table 5 lists information regarding tools and cutting conditions for machining alloy 390. The turning of alloy 390 can be performed using C-3 tungsten carbide or polycrystalline diamond

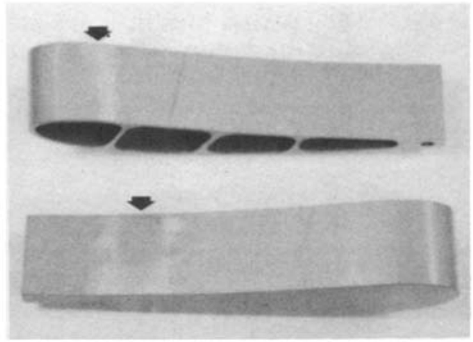


Fig. 12 Two views of a milling defect on an unsupported vertical flange in a 7075-T651 aluminum plate. Courtesy of McDonnell Douglas Canada, Ltd.

cutting tools. Turning with carbide should be limited to relatively slow cutting speeds (30 to 150 m/min, or 100 to 500 sfm) to ensure adequate tool life. Tool life can be greatly improved and cutting speeds substantially increased through the use of polycrystalline diamond tools. The productivity increase (300 to 900 m/min, or 1000 to 3000 sfm) can more than offset the higher cost of the diamond.

Similar results have been experienced for milling alloy 390. Milling with carbide should be done at slow speeds (99 to 175 m/min, or 325 to 575 sfm) with or without coolant. Milling with polycrystalline diamond can be done at speeds of 400 to 790 m/min (1300 to 2600 sfm) (with or without coolant) and with much longer tool life.

The drilling of alloy 390 can be accomplished using high-speed steel, high-helix drills with parabolic flutes at a speed of 20 m/min (70 sfm) for holes up to six times diameter in depth. Much better productivity can be achieved by using tungsten-carbide-tip, coolant-fed drills at 90 m/min (300 sfm). In addition, diamond-tip, coolant-fed gun drills have achieved even higher productivity gains with much better tool life in holes up to eight times diameter.

Thread forming taps have the longest tool life and can be run at speeds to 20 m/min (70 sfm). They also produce stronger threads than thread cutting taps.

The use of generous quantities of coolant is imperative when drilling and tapping alloy 390. The coolant must be in sufficient volume and pressure to reach the bottom of the hole and flush out the chips. These chips are very abrasive to the cutting tool.

The broaching of alloy 390 can be accomplished with the same speeds, tool geometries, and materials used to broach gray iron. Tool life will be much better, and the surface finish will be comparable to, or better than, the finish typical of gray iron.

The machining of alloy 390 in recent years has been much improved. This has been accomplished through the application of new technology in cutting tool materials and in tool designs, optimization of cutting

Table 7 Suggested practices for turning aluminum on general-purpose equipment

The suggested practices are commonly used, but higher speeds, feeds, and depths of cut can be employed in many cases (depending on the nature of the part, machine tool, tool design, lubrication, and other cutting conditions) to increase production rates.

Variable and condition	Suggested practice
Back rake angle, degrees	5–15
Side rake angle, degrees	
Machinability rating A	0–20
Machinability rating B	20
Machinability rating C	20–30
Machinability rating D, E	40
End relief angle clearance, degrees	
High-speed steel tool	8–15
Carbide tool	6–8
Side relief angle, degrees	
High-speed steel tool	8–15
Carbide tool	6–8
End cutting edge angle, degrees	5
Side cutting edge angle, degrees	5–15
Nose radius, mm (in.)	
Rough cutting	0.8–3.2 (1/32–1/8)
Finish cutting	0.25–0.76 (0.010–0.030)
Depth of cut, mm (in.)	
Small work	0.4–6.4 (1/64–1/4)
Large work	13–38 (1/2–1 1/2)
Feed, mm/rev (in./rev)	
Rough cutting	0.15–2.0 (0.006–0.080)
Finish cutting	0.05–0.15 (0.002–0.006)
Speed, m/min (sfm)	
High-speed steel tool	≤300 (≤1000)
Carbide tool	≤4600 (≤15 000)

Source: Ref 4

Table 8 Nominal speeds for turning aluminum alloys

Operation	Speed, m/min (sfm) Nonheat-treated cast alloys	All other alloys and tempers
High-speed steel tools (M2 or T5)		
Single-point roughing	230 (750)	180 (600)
Single-point finishing	300 (1000)	240 (800)
Forming and cutoff	170 (550)	140 (450)
Carbide tools (C-2)		
Single-point roughing(a)		
Brazed tips	490 (1600)	340 (1100)
Disposable tips	610 (2000)	430 (1400)
Forming and cutoff	360 (1200)	250 (825)

Note: Speeds for single-point turning are based on a 3.81 mm (0.150 in.) depth of cut and a feed of 0.38 mm/rev (0.015 in./rev) for roughing and a 0.64 mm (0.025 in.) depth of cut and a feed of 0.18 mm/rev (0.007 in./rev) for finishing. Speeds for form turning are based on feeds of 0.089, 0.08, and 0.05 mm/rev (0.0035, 0.003, and 0.002 in./rev) for tool widths of 13, 25, and 50 mm (1/2, 1, and 2 in.), respectively. Speeds for cutoff are based on a feed rate of 0.05 mm/rev (0.002 in./rev). (a) For finish turning, use the maximum speed of the machine and C-3 carbide.

conditions, and the replacement of outdated transfer lines with machine tools designed for aluminum alloys.

Cutting Fluids

A cutting fluid for aluminum can be a soluble-oil emulsion, a mineral oil, or an aqueous chemical solution. Cutting oils that contain compounds of sulfur or chlorine or both are seldom used and are not usually required for machining aluminum. In addition, many of them will stain the work. Table 6 lists various cutting fluids and their applications.

Soluble oil mixed with water in ratios of 1 part oil to 20 to 30 parts water is the cutting fluid most widely used for machining aluminum alloys. Soluble-oil emulsions are inexpensive, highly efficient for cooling and removing chips, and are usually adequate for preventing built-up edges.

Mineral oil used as a cutting fluid may contain a fatty additive, such as lard oil, neat's-foot oil, oleic acid, or butyl stearate. However, mineral oil that contains no additive and that has a viscosity of 40 to 300 Saybolt universal seconds (SUS) at 40 °C (100 °F) is most often used. As cutting speed is increased, the viscosity of the oil should be decreased to provide easier flow and therefore greater cooling. Straight mineral seal oil (~40 SUS at 40 °C, or 100 °F) has been effective in many applications. Kerosene, which is slightly less viscous than mineral seal oil, is also used.

Chemical solutions are effective as cutting fluids for machining aluminum and are especially desirable when a transparent fluid is needed to permit viewing the work during machining. These solutions vary in composition, but most contain amines, nitrides, phosphates, borates, soaps, wetting agents, glycols, and germicides. Some of these solutions stain aluminum alloys.

Stick grease is sometimes used in band sawing, circular sawing, and abrasive belt, abrasive disk, or abrasive wheel polishing and grinding when requirements are not too severe and a flood of lubricant is not required.

Continuous filtering of cutting fluids for the removal of chips, slivers, grindings, and other foreign material is especially desirable because aluminum alloys are relatively soft and are easily damaged by a contaminated cutting fluid.

The mist application of cutting fluids is sometimes preferred for machining aluminum, especially on such machine tools as small, vertical mills and high-speed routers. Compressed air, at pressures ranging from 70 to 550 kPa (10 to 80 psi) is used to atomize the cutting fluid and propel it into the cutting zone. The technique can be used with straight oils (especially of lower viscosities), emulsions, or chemical solutions as long as there are no solid additives contained in the fluid. Some producers of spray-mist equipment, however, recommend the use of water-base fluids over oils because the latter may tend to clog and may present a hazard.

Although the use of compressed air adds a new element of cost, the overall cost can often be reduced because of the smaller volume of fluid used and lower losses of fluid carried out by chips. The cooling effect

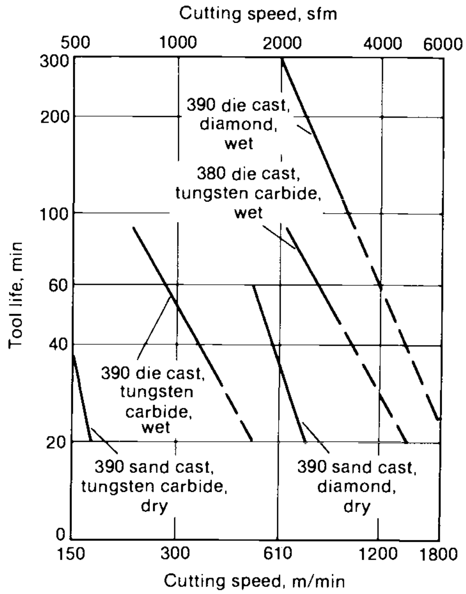


Fig. 14 Tool life curves for the dry turning of sand cast alloy 390 using carbide and diamond, the wet turning of die cast alloy 390 using carbide and diamond, and the wet turning of die cast alloy 380 using carbide. Source: Ref 2

of a mist, while not equal to that of a copious flood, is considerable because the rapid evaporation of the finely divided droplets adds the latent heat of vaporization to the cooling capability of the liquid.

Proponents of the compressed air method also claim that the high velocity of the droplets tends to get the fluid into the cutting zone effectively and that the evaporation of the water increases the lubricity at the cut. Visibility into the cutting area is improved, and splash shielding does not pose as much of a problem as flooding does. Most experts recommend the use of automatic shutoff devices for mist systems and venting to prevent potential air contamination.

Distortion and Dimensional Variation

Because aluminum alloys have a low modulus of elasticity (~70 GPa, or 10 × 10^6 psi), they will distort more than most metals for a given clamping or chucking force. Moderate clamping forces should be used to avoid dimensional variations due to distortion. High clamping pressures are not required, because cutting forces are low.

Thermal Expansion. The coefficient of thermal expansion of aluminum alloys (18 to 25 μm/m·K, or 10 to 14 μin./in./°F) is higher than that of most metals commonly machined. Therefore, the dimensional accuracy of finished parts requires that the part be kept cool during machining. When turning between centers, it is important to avoid expansion, which will put excessive pres-

Table 9 Nominal speeds and feeds for the boring of aluminum alloys with high-speed tool steels and carbide tools

					High-speed steel tool						Uncoated carbide tool							
									Tool material grade		Speed						Tool material grade	
	Hardness, HB (500 kg)		Depth of cut		Speed		Feed				Brazed		Indexable		Feed			
Material		Condition	mm	in.	m/min	sfm	mm/rev	in./rev	ISO	AISI	m/min	sfm	m/min	sfm	mm/rev	in./rev	ISO	C
Wrought																		
EC 3005 6066	30–80	Cold drawn	0.25	0.010	305	1000	0.075	0.003	S4, S5	M2, M3	610	2000	Max	Max	0.15	0.006	K01, M10	C-3
1060 4032 6070			1.25	0.050	245	800	0.13	0.005	S4, S5	M2, M3	550	1800	Max	Max	0.20	0.008	K10, M10	C-2
1100 5005 6101			2.5	0.100	215	700	0.30	0.012	S4, S5	M2, M3	365	1200	Max	Max	0.40	0.015	K20, M20	C-2
1145 5050 6151	75–150	Solution treated	0.25	0.010	305	1000	0.075	0.003	S4, S5	M2, M3	610	2000	Max	Max	0.15	0.006	K01, M10	C-3
1175 5052 6253		and aged	1.25	0.050	245	800	0.13	0.005	S4, S5	M2, M3	550	1800	Max	Max	0.20	0.008	K10, M10	C-2
2011 5083 6463			2.5	0.100	215	700	0.30	0.012	S4, S5	M2, M3	365	1200	Max	Max	0.40	0.015	K20, M20	C-2
1235 5056 6262																		
2014 5086 6951																		
2017 5154 7001																		
2018 5252 7004																		
2021 5254 7005																		
2024 5454 7039																		
2025 5456 7049																		
2117 5457 7050																		
2218 5652 7075																		
2219 5657 7079																		
2618 6053 7175																		
3003 6061 7178																		
3004 6063																		
Cast																		
Sand and permanent mold																		
A140 F332.0 520.0	40–100	As-cast	0.25	0.010	305	1000	0.075	0.003	S4, S5	M2, M3	550	1800	Max	Max	0.15	0.006	K01, M10	C-3
201.0 333.0 535.0			1.25	0.050	245	800	0.13	0.005	S4, S5	M2, M3	365	1200	Max	Max	0.20	0.008	K10, M10	C-2
208.0 354.0 705.0			2.5	0.100	215	700	0.30	0.012	S4, S5	M2, M3	305	1000	Max	Max	0.40	0.015	K20, M20	C-2
213.0 355.0 707.0	70–125	Solution treated	0.25	0.010	230	750	0.075	0.003	S4, S5	M2, M3	550	1800	Max	Max	0.15	0.006	K01, M10	C-3
222.0 C355.0 A712.0		and aged	1.25	0.050	185	600	0.13	0.005	S4, S5	M2, M3	365	1200	Max	Max	0.20	0.008	K10, M10	C-2
224.0 356.0 D712.0			2.5	0.100	170	550	0.30	0.012	S4, S5	M2, M3	305	1000	Max	Max	0.40	0.015	K20, M20	C-2
242.0 A356.0 713.0																		
295.0 357.0 771.0																		
B295.0 359.0 850.0																		
308.0 B443.0 A850.0																		
319.0 514.0 B850.0																		
328.0 A514.0																		
A332.0 B514.0																		
Hiduminium RR-350																		
Die castings																		
360.0 A380.0	40–100	As-cast	0.25	0.010	305	1000	0.075	0.003	S4, S5	M2, M3	550	1800	Max	Max	0.15	0.006	K01, M10	C-3
A360.0 C443.0			1.25	0.050	245	800	0.13	0.005	S4, S5	M2, M3	365	1200	Max	Max	0.20	0.008	K10, M10	C-2
380.0 518.0			2.5	0.100	215	700	0.30	0.012	S4, S5	M2, M3	305	1000	Max	Max	0.40	0.015	K20, M20	C-2
	70–125	Solution treated	0.25	0.010	230	750	0.075	0.003	S4, S5	M2, M3	490	1600	Max	Max	0.15	0.006	K01, M10	C-3
		and aged	1.25	0.050	185	600	0.13	0.005	S4, S5	M2, M3	425	1400	Max	Max	0.20	0.008	K10, M10	C-2
			2.5	0.100	145	550	0.30	0.012	S4, S5	M2, M3	245	800	Max	Max	0.40	0.015	K20, M20	C-2
383.0 413.0	40–100	As-cast	0.25	0.010	230	750	0.075	0.003	S4, S5	M2, M3	490	1600	Max	Max	0.15	0.006	K01, M10	C-3
A384.0 A413.0			1.25	0.050	185	600	0.13	0.005	S4, S5	M2, M3	425	1400	Max	Max	0.20	0.008	K10, M10	C-2
			2.5	0.100	170	550	0.30	0.012	S4, S5	M2, M3	245	800	Max	Max	0.40	0.015	K20, M20	C-2
	70–125	Solution treated	0.25	0.010	205	675	0.075	0.003	S4, S5	M2, M3	410	1350	Max	Max	0.15	0.006	K01, M10	C-3
		and aged	1.25	0.050	170	550	0.13	0.005	S4, S5	M2, M3	335	1100	Max	Max	0.20	0.008	K10, M10	C-2
			2.5	0.100	145	480	0.30	0.012	S4, S5	M2, M3	260	850	Max	Max	0.40	0.015	K20, M20	C-2
390.0	40–100	As-cast	0.25	0.010	46	150	0.075	0.003	S4, S5	M2, M3	125	410	145	480	0.15	0.006	K10, M10	C-2
392.0			1.25	0.050	37	120	0.13	0.005	S4, S5	M2, M3	100	325	115	385	0.20	0.008	K10, M10	C-2
			2.5	0.100	34	110	0.30	0.012	S4, S5	M2, M3	90	300	105	350	0.30	0.012	K20, M20	C-2
	70–125	Solution treated	0.25	0.010	43	140	0.075	0.003	S4, S5	M2, M3	105	350	125	410	0.15	0.006	K10, M10	C-2
		and aged	1.25	0.050	34	110	0.13	0.005	S4, S5	M2, M3	85	280	100	330	0.20	0.008	K10, M10	C-2
			2.5	0.100	30	100	0.30	0.012	S4, S5	M2, M3	70	230	84	275	0.30	0.012	K20, M20	C-2

Source: Ref 5

sure on the centers. High cutting speed helps keep the part cool because most of the heat introduced into the part during a given rotation is removed with the chip during the next rotation, and the time for diffusion of heat is short. A cutting fluid is effective in removing heat that is not removed with the chips. Live centers are recommended to minimize frictional heating at the center. Dull tools cause a heat rise in the workpiece; therefore, cutting tools should be kept sharp.

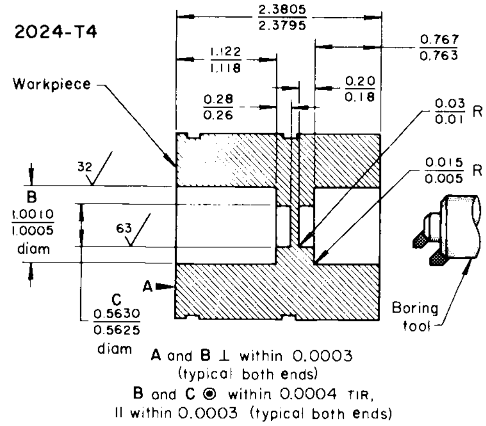
Residual stress can be induced by a dull or improperly designed cutter that cold works the surface, by excessive chucking or clamping force, or by faulty clamping. Distortion from residual stress is most noticeable in slender parts.

The distortion resulting from machining stresses can be minimized or eliminated either by employing a series of light cuts as the part approaches finished size or by stress relieving the part between rough and finish machining. For heat-treatable alloys, it is preferable to do all rough machining on material in the solution-treated and aged condition, rather than in the annealed con-

Table 10 Nominal speeds and feeds for the boring of aluminum alloys with diamond tools

Material	Hardness, HB (500 kg)	Condition	Depth of cut, mm (in.)	Speed, m/min (sfm)	Feed, mm/rev (in./rev)
Wrought					
EC 2218 5252 6253	30–150	All	0.13–0.40	460	0.075–0.15
1060 2219 5254 6262			(0.005–0.015)	(1500)	(0.003–0.006)
1100 2618 5454 6463			0.40–1.25	305	0.15–0.30
1145 3003 5456 6951			(0.015–0.050)	(1000)	(0.006–0.012)
1175 3004 5457 7001			1.25–3.2	150	0.30–0.50
1235 3005 5652 7004			(0.050–0.125)	(500)	(0.012–0.020)
2011 4032 5657 7005					
2014 5005 6053 7039					
2017 5050 6061 7049					
2018 5052 6063 7050					
2021 5056 6066 7075					
2024 5083 6070 7079					
2025 5086 6101 7175					
2117 5154 6151 7178					
Cast					
Sand and permanent mold					
A140 319.0 357.0 A712.0	40–100	As-cast	0.13–0.40	760	0.075–0.15
201.0 328.0 359.0 D712.0			(0.005–0.015)	(2500)	(0.003–0.006)
208.0 A332.0 B443.0 713.0			0.40–1.25	610	0.15–0.30
213.0 F332.0 514.0 771.0			(0.015–0.050)	(2000)	(0.006–0.012)
222.0 333.0 A514.0 850.0			1.25–3.2	305	0.30–0.50
224.0 354.0 B514.0 A850.0			(0.050–0.125)	(1000)	(0.012–0.020)
242.0 355.0 520.0 B850.0	70–125	Solution treated and aged	0.13–0.40	670	0.075–0.15
295.0 C355.0 535.0			(0.005–0.015)	(2200)	(0.003–0.006)
B295.0 356.0 705.0			0.40–1.25	520	0.15–0.30
308.0 A356.0 707.0			(0.015–0.050)	(1700)	(0.006–0.012)
Hiduminium RR-350			1.25–3.2	215	0.30–0.50
			(0.050–0.125)	(700)	(0.012–0.020)
Die castings					
360.0 A380.0	40–100	As-cast	0.13–0.40	915	0.075–0.15
A360.0 C443.0			(0.005–0.015)	(3000)	(0.003–0.006)
380.0 518.0			0.40–1.25	760	0.15–0.30
383.0			(0.015–0.050)	(2500)	(0.006–0.012)
			1.25–3.2	610	0.30–0.50
			(0.050–0.125)	(2000)	(0.012–0.020)
A384.0	40–100	As-cast	0.13–0.40	760	0.075–0.15
413.0			(0.005–0.015)	(2500)	(0.003–0.006)
A413.0			0.40–1.25	610	0.15–0.30
390.0			(0.015–0.050)	(2000)	(0.006–0.012)
			1.25–3.2	460	0.30–0.50
			(0.050–0.125)	(1500)	(0.012–0.020)
392.0	40–100	As-cast	0.13–0.40	610	0.075–0.15
			(0.005–0.015)	(2000)	(0.003–0.006)
			0.40–1.25	460	0.15–0.30
			(0.015–0.050)	(1500)	(0.006–0.012)
			1.25–3.2	305	0.30–0.50
			(0.050–0.125)	(1000)	(0.012–0.020)

Source: Ref 5



Speed, at 1200 rev/min, m/min (sfm)	97 (318)
Feed, mm/min (in./min)	35.6 (1.4)
Cutting fluid	Kerosene plus proprietary oil
Production rate, pieces/h	9
Tool life/grind, lineal m (lineal in.)	20 (800)

Fig. 15 Rough and finish boring of a heat-treated aluminum alloy forging. Dimensions in figure given in inches

dition, because the less ductile structure is more machinable.

The distortion resulting from machining stress can often be minimized by purchasing the alloy in a stress-relieved condition, normally designated by T451, T651, or T851 if the metal has been stress relieved by stretching. The code T*x*52 denotes stress relief by compression; T*x*53, stress relief by heat treating.

A major source of dimensional variation arises from the presence of movement, or play, in the feed mechanism of the machine. When the tool or the machining conditions cause forces that take up the play completely, no variations in dimensions are encountered. However, when low cutting forces (typical of those required for machining aluminum) are combined with a small feed and light cut, the total force may be insufficient to overcome tool-slide friction. Then, some movement may still occur, and the tool may float.

Soft Spots in Machined Aluminum Parts (Ref 3)

Machining practices that are less than optimum may cause localized overheating in aluminum alloys. In alloys being machined in the postsolution heat treatment tempers, the resultant effect is overaging, softening, and therefore loss of strength.

Such defects, referred to as soft spots, are usually invisible and may remain undetected at this stage except where hardness and/or conductivity testing is performed in the overheated area. However, anodizing has the capability to reveal these defects. Locally overheated and softened areas are indicated by a color contrast against the background of the normal anodic film. The area is always very local on the parts, and its shape is usually related to the machine cutting tool that profiled the particular area. Frequently, the soft areas are at or near changes of section thickness. In addition, there is often exact repeatability of the soft spot location in all other parts of the batch.

Figure 6 illustrates a half-moon defect associated with spar milling deficiencies in an area adjacent to a change in section thickness. Figure 7 shows a circular indication that is associated with vertical plunge cut pocketing operations using an end mill. The 25 mm (1 in.) diameter of the soft spot is a function of the end mill diameter. Figure 8 shows a similar example of a milling problem that occurred in a numerically controlled (NC) machining application using a 75 mm (3 in.) diam cutter.

A different type of problem is illustrated in Fig. 9. Two silver-white defective areas are clearly visible on the tee section part machined from an extrusion. All 36 parts in the batch were similarly affected. The defects are attributed to a lack of full-support tooling backup that did not provide an adequate heat sink. The milling cutter was allowed to dwell at the start of the rebate cut, thus introducing frictional heating. The

Table 11 Nominal speeds and feeds for the planing of aluminum alloys with high-speed tool steels and carbide tools

				High-speed steel tool			Carbide tool		
Material	Hardness, HB (500 kg)	Condition	Depth of cut, mm (in.)	Speed, m/min (sfm)	Feed, mm/stroke (in./stroke)	Tool material grade, ISO (AISI)	Speed, m/min (sfm)	Feed, mm/stroke (in./stroke)	Tool material grade, ISO (C grade)
Wrought									
EC 2218 5252 6253	30–80	Cold drawn	0.25	90	(a)	S4, S5	90	(a)	K10, M10
1060 2219 5254 6262			(0.010)	(300)	(a)	(M2, M3)	(300)	(a)	(C-2)
1100 2618 5454 6463			2.5	90	3.20	S4, S5	90	3.20	K20, M20
1145 3003 5456 6951			(0.100)	(300)	(0.125)	(M2, M3)	(300)	(0.125)	(C-2)
1175 3004 5457 7001			12	90	2.30	S4, S5	90	2.30	K30, M30
1235 3005 5652 7004			(0.500)	(300)	(0.090)	(M2, M3)	(300)	(0.090)	(C-2)
2011 4032 5657 7005	75–150	Solution treated and aged	0.25	90	(a)	S4, S5	90	(a)	K10, M10
2014 5005 6053 7039			(0.010)	(300)	(a)	(M2, M3)	(300)	(a)	(C-2)
2017 5050 6061 7049			2.5	90	3.20	S4, S5	90	3.20	K20, M20
2018 5052 6063 7050			(0.100)	(300)	(0.125)	(M2, M3)	(300)	(0.125)	(C-2)
2021 5056 6066 7075			12	90	2.30	S4, S5	90	2.30	K30, M30
2024 5083 6070 7079			(0.500)	(300)	(0.090)	(M2, M3)	(300)	(0.090)	(C-2)
2025 5086 6101 7175									
2117 5154 6151 7178									
Cast									
Sand and permanent mold									
A140 319.0 357.0 A712.0	40–100	As-cast	0.25	90	(a)	S4, S5	90	(a)	K10, M10
201.0 328.0 359.0 D712.0			(0.010)	(300)	(a)	(M2, M3)	(300)	(a)	(C-2)
208.0 A332.0 B443.0 713.0			2.5	90	3.20	S4, S5	90	3.20	K20, M20
213.0 F332.0 514.0 771.0			(0.100)	(300)	(0.125)	(M2, M3)	(300)	(0.125)	(C-2)
222.0 333.0 A514.0 850.0			12	90	2.30	S4, S5	90	2.30	K30, M30
224.0 354.0 B514.0 A850.0			(0.500)	(300)	(0.040)	(M2, M3)	(300)	(0.090)	(C-2)
242.0 355.0 520.0 B850.0	70–125	Solution treated and aged	0.25	90	(a)	S4, S5	90	(a)	K10, M10
295.0 C355.0 535.0			(0.010)	(300)	(a)	(M2, M3)	(300)	(a)	(C-2)
B295.0 356.0 705.0			2.5	90	3.20	S4, S5	90	3.20	K20, M20
308.0 A356.0 707.0			(0.100)	(300)	(0.125)	(M2, M3)	(300)	(0.125)	(C-2)
Hiduminium RR-350			12	90	2.30	S4, S5	90	2.30	K30, M30
			(0.500)	(300)	(0.090)	(M2, M3)	(300)	(0.090)	(C-2)

(a) Feed = ¾ the width of square nose finishing tool. Source: Ref 5

part was then turned over, and the operation was repeated, thus producing an opposite soft spot. The milling sequence is outlined in Fig. 10. This illustrates clearly the phenomenon of soft spot location repeatability.

Another example of soft spot indications is shown in Fig. 11. Of the 19 parts produced in this batch, 13 were affected. The defects are attributed to vibration or distortion at an inadequately clamped edge of the parts during milling on a vacuum chuck. As a result, heat dissipation was restricted.

All soft spots have a lower hardness and a higher electrical conductivity than the adjacent normal material. The strength reduction can be significant.

Sources of Machining Problems

Some of the more common machining problems are outlined in this section. It should be kept in mind that the basic cause of the defects under discussion is always excessive heat. This may arise directly by frictional heating and/or indirectly by insufficient cooling. Because the cause(s) of some problems may be obscure, on-site analysis of the hardware, machining techniques, tooling, and so on, is always desirable.

Cutters. All tools should be kept sharp and in good condition at all times. Tool geometry must be maintained within the established requirements for aluminum alloys.

Cutters must have the optimum number of flutes and the optimum spiral configuration for each application. For most applications, the minimum number of flutes and a coarse spiral are more desirable. Clogging results in rubbing and frictional heating; it can be minimized by maintaining a good surface finish in the flutes.

Feeds and speeds must be correct for the operation being performed. In particular, a slow feed rate (dwelling) combined with high spindle speeds is especially troublesome. Great care must be taken not to reduce feeds any more than is absolutely necessary when the cutter is approaching or is actually machining areas requiring section changes. On multispindle equipment, it is vital that sufficient net horsepower be available to each spindle during heavy roughing cuts. Where practical, when using end mills, deep plunge cuts should ramp in or start from a predrilled hole and should progress on a multipass basis.

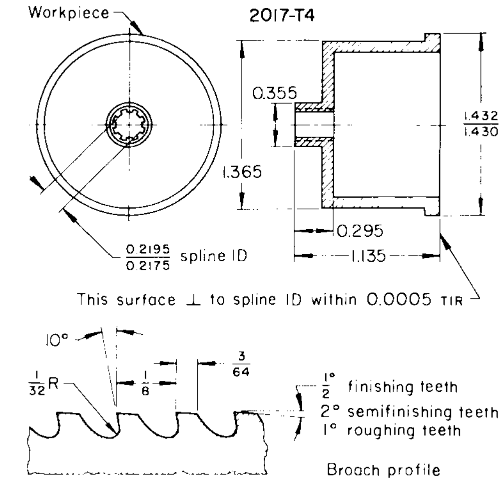
Cutting Fluids. An adequate and continuous flow of cutting fluid (flood or mist) directed at the cutting edges is essential. The flow of cutting fluid should begin before cutting and must continue until the cutter has been removed from the part. Fluid circulation must be continuous because even a momentary reduction in flow, such as could be caused by an air lock, may result in local overheating.

Support Fixtures and Clamping Methods. Machining backup fixtures must be rigid and must have sufficient contact area and mass to provide an adequate heat absorption capacity. Gaps in modular fixtures should not be excessive. This is especially true in areas adjacent to section changes. In addition, it is vital that clamping methods maintain continuous contact between part and fixtures because even a minute air gap due to part distortion or vibration, together with the resultant deeper cutting action, may produce overheated areas. On flat parts that are being machined on a vacuum chuck, it is important that the edge seal be located as close to the periphery as practical in order to eliminate vibration and consequent gapping around the outer edges. In general, all thin edges are susceptible to heat buildup. Similarly, thin-section vertical

Table 12 Nominal speeds and feeds per tooth (chip load) for the broaching of aluminum alloys with high-speed tool steels

Material				Hardness, HB (500 kg)	Condition	Speed, m/min (sfm)	Chip load, mm/tooth (in./tooth)	Tool material grade, ISO (AISI or C)
Wrought								
EC	2218	5252	6253	30–80	Cold drawn	12 (40)	0.15 (0.006)	S4, S2 (M2, M7)
1060	2219	5254	6262	75–150	Solution treated	12 (40)	0.15 (0.006)	S4, S2 (M2, M7)
1100	2618	5454	6463		and aged			
1145	3003	5456	6951					
1175	3004	5457	7001					
1235	3005	5652	7004					
2011	4032	5657	7005					
2014	5005	6053	7039					
2017	5050	6061	7049					
2018	5052	6063	7050					
2021	5056	6066	7075					
2024	5083	6070	7079					
2025	5086	6101	7175					
2117	5154	6151	7178					
Cast								
Sand and permanent mold								
A140	319.0	357.0	A712.0	40–100	As-cast	12 (40)	0.15 (0.006)	S4, S2 (M2, M7)
201.0	328.0	359.0	D712.0	70–125	Solution treated	12 (40)	0.15 (0.006)	S4, S2 (M2, M7)
208.0	A332.0	B443.0	713.0		and aged			
213.0	F332.0	514.0	771.0					
222.0	333.0	A514.0	850.0					
224.0	354.0	B514.0	A850.0					
242.0	355.0	520.0	B850.0					
295.0	C355.0	535.0						
B295.0	356.0	705.0						
308.0	A356.0	707.0						
Hiduminium RR-350								

Source: Ref 5



Speed, m/min (sfm)	9.1 (30)
Feed, mm/tooth (in./tooth)	0.05 (0.002)
Length of stroke, mm (in.)	152 (6)
Cutting fluid	Soluble oil:water (1:25)
Setup time, min	30
Production rate, pieces/h	180
Tool life/grind	2000 pieces
Total tool life	40 000 pieces

Fig. 16 Clutch housing and details of broach used for producing the internal splines. Dimensions in figure given in inches

webs of pocketed or stiffened parts, for which no backup support is practical, are also prone to these problems. Figure 12 illustrates this latter problem on a small pocketed end fitting in which the effect is noticeable on the vertical flanges.

Numerically Controlled Machining Operations. All of the foregoing considerations regarding sources of machining problems also apply to NC operations. In addition, however, there is generally less operator involvement or awareness of the actual machining operation in NC processes. For example, in some plants, one operator may attend several machines. Futhermore, the programmer may be less aware of possible pitfalls, and the optimum mathematical cutter path may create local potential problems. Dwelling at corners or between subroutines must be strictly avoided.

Quality Control. The causes of overheating during machining are well recognized (Fig. 13). However, the safety margin between acceptable and unacceptable practices is not wide; therefore, the utmost care and attention to detail is warranted at all times.

The maximum impact of these problems obviously occurs over the short term. The tendency of these defects to be repetitive in one batch of parts can raise havoc with a tight production schedule if none of the parts is acceptable. Wholesale rejections can be avoided by close monitoring of material conductivity values in critical areas during machining, particularly on new parts. As a result, the impact of downstream economic losses and potential major operations difficulties (cost of replacement parts plus the associated disruptive factors) can be minimized.

Turning

As indicated in Fig. 2 and 3, the cutting force, and therefore the power, required to turn aluminum is considerably less than that needed to turn low-carbon steel of approximately the same hardness and tensile properties.

Tool Design. The recommended angles for a single-point lathe tool are given in Table 7. Cutting angles should be on the high end of the ranges for alloys that are not heat treatable or that are more ductile than the free-cutting alloys.

Low modulus of elasticity, high ductility, and the tendency to stick to tool materials are the principal factors dictating differences in the cutting tools used on aluminum. Because of its low modulus of elasticity, aluminum has a greater tendency to yield, or spring, under the pressure of a cutting tool. It deforms more than steel before the cutter can start to bite. For this reason, a fine cutting edge is desirable, and higher clearance angles are generally recommended—8 to 15° for high-speed steel tools, 6 to 10° for carbides, and somewhat less for diamonds.

Traditional practice in the machining of aluminum has specified high, positive rake angles, both back and side, for high-speed steel and carbide tools, especially for the gummier alloys rated D and E in machinability. Although these recommendations are specifically for single-point turning operations, they are also generally applicable to other types of tools. In addition, as the alloy in question increases in Brinell hardness, in alloying complexity, and in machinability rating, lower rake angles can be used.

One of the major reasons for using high-rake tooling is to prevent, or at least delay, the formation of a built-up edge on the cutter. For the same reason and to facilitate chip flow, it is universally recommended that all tool surfaces that contact either the workpiece or the chip be given a high polish. Any grinding marks on the rake face or flank of the tool should be parallel to the direction of the chip-flow or the cut, whichever is applicable. Grinding marks should be removed by stoning or polishing.

Because carbide tools are more brittle than high-speed steel tools, they will chip or break when the cutting angles are too large. In turning free-cutting alloys, increasing the side relief or side rake angle, or both, will reduce the power required. Because the power needed for turning aluminum alloys

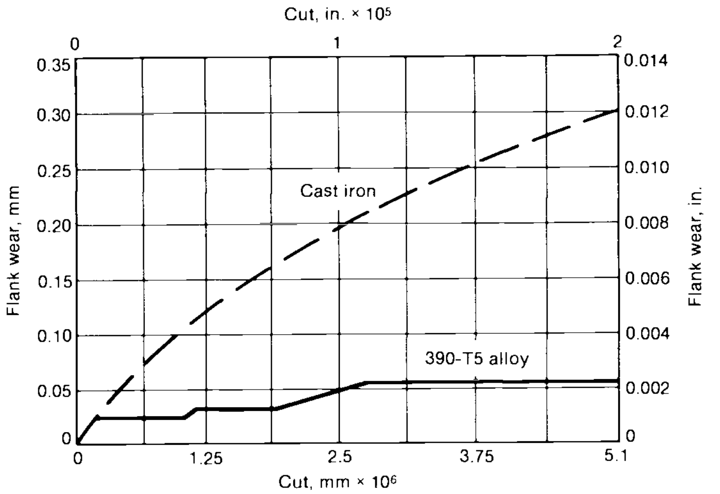


Fig. 17 Typical tool wear when broaching with a carbide cutting tool at 45 m/min (150 sfm). Source: Ref 2

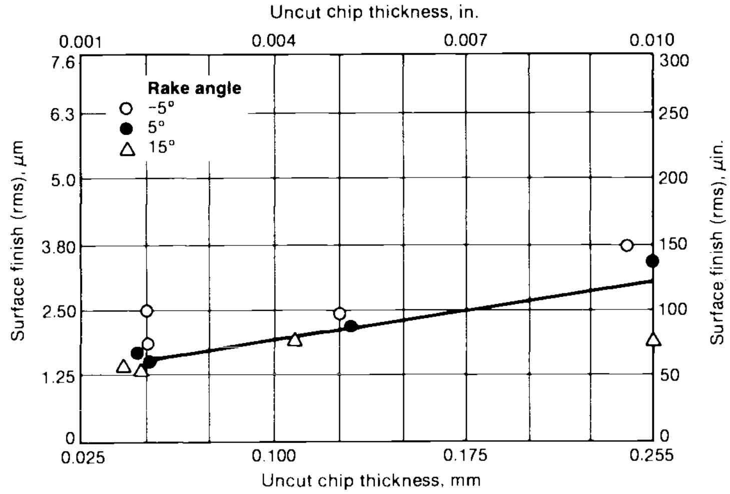


Fig. 18 Surface finish at various depths of cut when broaching 390 aluminum alloy of 100 HB at 45 m/min (150 sfm) with a carbide tool. Although the rake angle varied, the shear angle of the tool was 0° in all three cases. Source: Ref 2

is small (about one-third that needed for machining soft low-carbon steel), a turning tool ground with angles that are too large is likely to hog the work metal. Because the tool floats in operation, any backlash or play in the spindle or machine ways causes difficulty in holding tolerances. Tool life and surface finish also suffer. Surface finish can be improved by grinding a large nose radius on the turning tool. However, the maximum nose radius depends partly on the allowable fillet on the workpiece when turning to a shoulder. As the nose radius is made larger, cutting force and horsepower requirements increase. On small-diameter stock, increased cutting force may cause the work metal to bend away from the tool; therefore, stock supports or rests are required. A nose radius that is too large also causes chatter, which results in poor surface finish and possibly tool breakage.

Another way to improve surface finish is to grind the end cutting edge of the tool parallel to the work for a width equal to 1½ to 2 times the feed rate in millimeters per revolution. This flat edge will cut behind the nose of the tool to smooth out the ridges caused by the feed. Too wide a flat will cause chatter, and a negative angle will leave a taper equal in length to the width of the flat, on the work at the end of the turn.

Tool Honing. Honing a carbide tool with a diamond-impregnated stone will improve the surface finish on the workpiece and will extend the life of the tool. Disposable carbide inserts have a honed edge when purchased, and no additional honing is necessary.

Tool Material. Carbide tools, either brazed-insert or disposable-insert, last far longer than high-speed steel tools for turning any aluminum alloy. For turning the high-silicon alloys, the use of carbide tools or polycrystalline diamond is mandatory for optimum results (see the section "Machining of High-Silicon Aluminum Alloy 390" in this article).

The speed used in turning aluminum alloys depends to some extent on the alloy and condition, but far more on tool material and type of tool (single-point, form, or cutoff). The effect of alloy composition and condition is small in selecting a turning speed, except for the nonheat-treated cast alloys.

Nominal speeds for turning aluminum with high-speed steel and carbide tools are given in Table 8. These speeds are based on feeds and depths of cut that are typical for turning aluminum alloys and on the assumptions that the setup is rigid, that a cutting fluid is used, and that the workpiece can be rotated to attain these surface speeds without causing excessive vibration. When conditions are below normal, speeds must be scaled down from those given in Table 8. Similarly, under nearly ideal conditions, higher speeds are often used successfully.

Feed. For rough turning with single-point tools, regardless of tool material, a feed of 0.38 mm/rev (0.015 in./rev) is common for all aluminum alloys. For finish turning, a feed of 0.18 mm/rev (0.007 in./rev) is recommended; this feed will usually result in a surface finish of 1.60 to 3.20 µm (63 to 125 µin.). Lighter feeds are sometimes used to provide a better finish.

In form turning, the width of the form tool is the major variable that affects the rate of feed. For all alloys and tool materials, a feed rate of 0.089 mm/rev (0.0035 in./rev) is generally satisfactory for form tools no wider than 12.7 mm (0.500 in.). Rate of feed should be decreased as the width of the form tool increases. Feed rates of 0.08 and 0.05 mm/rev (0.003 and 0.002 in./rev) are recommended for form tool widths of 25 and 50 mm (1 and 2 in.), respectively.

For cutoff tools, a feed rate of 0.05 mm/rev (0.002 in./rev) is usually satisfactory, regardless of the alloy being turned, the tool material, or the width of the cutoff tool.

Depth of cut in turning aluminum often depends on the power available. The speeds given in Table 7 are based on a 3.81 mm (0.150 in.) depth of cut for roughing and 0.64 mm (0.025 in.) for finishing. When power is available, roughing cuts of 6.35 mm (0.250 in.) are common. Finishing cuts of less than 0.64 mm (0.025 in.) often result in better surface finish.

Cutting fluid is recommended for turning all aluminum alloys (see the section "Cutting Fluids" in this article).

The procedures and equipment for turning that are the same for all metals are discussed in the article "Turning" in this Volume (special considerations for high-silicon content alloys are given below). Because aluminum alloys are far less sensitive to abrupt changes in speed, feed, and depth

Table 13 Tool angles for drilling aluminum alloys with high-speed steel drills in drill presses

Point angle (θ), degrees	
Thin stock	118–150
Thinned point	(a)
General work	118–140
Aluminum-silicon alloys	≥90
Helix angle, degrees	
Thin stock	24
Medium depth(b)	28
Deep holes(c)	40–48
Lip clearance, degrees	
Soft alloys	17
Strong alloys	15
Aluminum-silicon alloys	12

(a) Diameter ÷ 1.8 stock thickness = tan θ/2. (b) Up to six times drill diameter. (c) Over six times drill diameter

Table 14 Nominal speeds and feeds for the drilling of aluminum alloys with high-speed tool steel tool

Material	Hardness, HB (500 kg)	Condition	Speed, m/min (sfm)	Feed, mm/rev (in./rev)(a) at a nominal hole diameter of: 1.5 mm (1/16 in.)	3 mm (1/8 in.)	6 mm (1/4 in.)	12 mm (1/2 in.)	18 mm (3/4 in.)	25 mm (1 in.)	35 mm (1 1/2 in.)	50 mm (2 in.)	Tool material grade, ISO (AISI or C)
Wrought												
EC 3005 6066	30–80	Cold drawn	43	0.025	...	...	...	...	...	...	...	S2, S3
1060 4032 6070			(140)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
1100 5005 6101			105	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
1145 5050 6151			(350)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)
1175 5052 6253	75–150	Solution treated	43	0.025	...	...	...	...	...	...	...	S2, S3
1235 5056 6262		and aged	(140)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
2011 5083 6463			84	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
2014 5086 6951			(275)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)
2017 5154 7001												
2018 5252 7004												
2021 5254 7005												
2024 5454 7039												
2025 5456 7049												
2117 5457 7050												
2218 5652 7075												
2219 5657 7079												
2618 6053 7175												
3003 6061 7178												
3004 6063												
Cast												
Sand and permanent mold												
A410 F332.0 520.0	40–100	As-cast	43	0.025	...	...	...	...	...	...	...	S2, S3
201.0 333.0 535.0			(140)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
208.0 354.0 705.0			105	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
213.0 355.0 707.0			(350)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)
222.0 C355.0 A712.0	70–125	Solution treated	43	0.025	...	...	...	...	...	...	...	S2, S3
224.0 356.0 D712.0		and aged	(140)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
242.0 A356.0 713.0			84	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
295.0 357.0 771.0			(275)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)
B295.0 359.0 850.0												
308.0 B443.0 A850.0												
319.0 514.0 B850.0												
328.0 A514.0												
A332.0 B514.0												
Hiduminium RR-350												
Die castings												
360.0 A380.0	40–100	As-cast	46	0.025	...	...	...	...	...	...	...	S2, S3
A360.0 C443.0			(150)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
380.0 518.0			115	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
			(375)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)
	70–125	Solution treated	46	0.025	...	...	...	...	...	...	...	S2, S3
		and aged	(150)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
			90	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
			(300)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)
383.0 413.0	40–100	As-cast	30	0.025	...	...	...	...	...	...	...	S2, S3
A384.0 A413.0			(100)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
			43	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
			(140)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)
	70–125	Solution treated	27	0.025	...	...	...	...	...	...	...	S2, S3
		and aged	(90)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
			37	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
			(120)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)
390.0	40–100	As-cast	8	0.025	...	...	...	...	...	...	...	S2, S3
392.0			(25)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
			15	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
			(50)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)
	70–125	Solution treated	8	0.025	...	...	...	...	...	...	...	S2, S3
		and aged	(25)	(0.001)	...	...	...	...	...	...	...	(M10, M7, M1)
			14	...	0.075	0.18	0.30	0.40	0.50	0.65	0.75	S2, S3
			(45)	...	(0.003)	(0.007)	(0.012)	(0.016)	(0.019)	(0.025)	(0.030)	(M10, M7, M1)

(a) For holes more than two diameters deep, reduce speed and feed. Source: Ref 5

of cut than many other metals, they are especially well adapted to turning in automatic equipment.

The turning of high-silicon aluminum alloys (Ref 2) such as alloy 390 is best accomplished with polycrystalline diamond tools at speeds up to 900 m/min (3000 sfm). Carbide tools are satisfactory for speeds up to 150 m/min (500 sfm). Carbide tools should be polished for best results. Either tool material should have at least a slight positive rake. The use of a coolant will always improve tool life; however, alloy 390 can be turned dry, if required, but with somewhat reduced tool life. Dry turning speeds should be limited to 300 m/min (1000 sfm) for diamond and 30 m/min (100 sfm) for carbide tools.

Table 15 Nominal speeds and feeds for the spade drilling of aluminum alloys with high-speed tool steels and carbide tools

Material			Hardness, HB (500 kg)	Condition	Speed, m/min (sfm)	Feed, mm/rev (in./rev), at a nominal hole diameter of: 18–25 mm (¾–1 in.)	35 mm (1¼ in.)	50 mm (2 in.)	75 mm (3 in.)	100 mm (4 in.)	150 mm (6 in.)	200 mm (8 in.)	200 mm and up (8 in. and up)	Tool material grade, ISO (AISI or C)
Wrought														
EC	3005	6066	30–80	Cold drawn	90	0.40	0.45	0.65	0.90	1.10	1.65	1.90	2.30	S3, S4, S5, S2
1060	4032	6070			(300)	(0.015)	(0.018)	(0.026)	(0.035)	(0.042)	(0.065)	(0.075)	(0.090)	(M1, M2, M3, M10)
1100	5005	6101			185	0.13	0.15	0.20	0.30	0.36	0.55	0.65	0.75	K20
1145	5050	6151			(600)	(0.005)	(0.006)	(0.008)	(0.012)	(0.014)	(0.022)	(0.025)	(0.030)	(C-2)
1175	5052	6253	75–150	Solution treated	84	0.40	0.45	0.65	0.90	1.10	1.65	1.90	2.30	S3, S4, S5, S2
1235	5056	6262		and aged	(275)	(0.015)	(0.018)	(0.026)	(0.035)	(0.042)	(0.065)	(0.075)	(0.090)	(M1, M2, M3, M10)
2011	5083	6463			185	0.13	0.15	0.20	0.30	0.36	0.55	0.65	0.75	K20
2014	5086	6951			(600)	(0.005)	(0.006)	(0.008)	(0.012)	(0.014)	(0.022)	(0.025)	(0.030)	(C-2)
2017	5154	7001												
2018	5252	7004												
2021	5254	7005												
2024	5454	7039												
2025	5456	7049												
2117	5457	7050												
2218	5652	7075												
2219	5657	7079												
2618	6053	7175												
3003	6061	7178												
3004	6063													
Cast														
Sand and permanent mold														
A140	F332.0	520.0	40–100	As-cast	90	0.40	0.45	0.65	0.90	1.10	1.65	1.90	2.30	S3, S4, S5, S2
201.0	333.0	535.0			(300)	(0.015)	(0.018)	(0.026)	(0.035)	(0.042)	(0.065)	(0.075)	(0.090)	(M1, M2, M3, M10)
208.0	354.0	705.0			185	0.13	0.15	0.20	0.30	0.36	0.55	0.65	0.75	K20
213.0	355.0	707.0			(600)	(0.005)	(0.006)	(0.008)	(0.012)	(0.014)	(0.022)	(0.025)	(0.030)	(C-2)
222.0	C355.0	A712.0	70–125	Solution treated	84	0.40	0.45	0.65	0.90	1.10	1.65	1.90	2.30	S3, S4, S5, S2
224.0	356.0	D712.0		and aged	(275)	(0.015)	(0.018)	(0.026)	(0.035)	(0.042)	(0.065)	(0.075)	(0.090)	(M1, M2, M3, M10)
242.0	A356.0	713.0			185	0.13	0.15	0.20	0.30	0.36	0.55	0.65	0.75	K20
295.0	357.0	771.0			(600)	(0.005)	(0.006)	(0.008)	(0.012)	(0.014)	(0.022)	(0.025)	(0.030)	(C-2)
B295.0	359.0	850.0												
308.0	B443.0	A850.0												
319.0	514.0	B850.0												
328.0	A514.0													
A332.0	B514.0													
Hiduminium RR-350														

Source: Ref 5

High-speed steel tools yield consistently poor results. Tool wear is rapid, and workpiece buildup on the cutting tool is rapid and inconsistent, resulting in poor surface finish and difficult-to-control dimensions.

Ceramic tools exhibit rapid tool edge buildup at high speeds with consequent loss of finish and geometry. At lower speeds, these conditions improve, but excessive heat transmitted to the ceramic material results in premature failure due to fracture of the tool insert.

The best overall carbide grade has been found to be C-3, with C-2 as next best. Both are typical grades used for machining gray iron. It is desirable to have a good polish on the top rake face of the tool to minimize tool edge buildup. Any good natural or synthetic soluble-oil emulsion is recommended in order to prolong tool life. Generally, positive rake and relief angles and a sharp nose radius promote good surface finish and minimize tool edge buildup for both carbide and polycrystalline diamond tools. Tool grades and geometry for turning 390 aluminum alloy are:

Tools	Tungsten carbide, C-3, SPG422 insert with J polish
	Polycrystalline diamond, SPG422 insert
Geometry	0° Back rake angle
	5° Side rake angle
	5° Side rake angle
	5° End clearance angle (relief)
	5° Side clearance angle (relief)
	15° Minor cutting edge angle (end)
	15° Lead angle (side)
	0.76 mm (0.030 in.) corner radius

Polycrystalline diamond is destined to become the most cost-effective tool material for machining harder and more abrasive casting alloys such as 390. It can ensure a potential improvement in transfer line productivity.

Tool lives for the machining of alloys 380 and 390 are shown in Fig. 14. The data represent the cutting times required to achieve a 0.38 mm (0.015 in.) uniform wear land on the flank face of the cutting tool (except for the die cast alloy 390 machined with diamond; in this case, cutting was stopped at a uniform wear land of 0.25 mm, or 0.010 in.; and the 0.38 mm, or 0.015 in., tool life would be even greater, to the right in Fig. 14). All of the turning tests, except for sand cast alloy 390, were conducted using soluble-oil emulsion coolant at the ratio of 20:1. The dashed lines in Fig. 14 represent extrapolations of actual data (solid lines).

Boring

The boring of aluminum alloys, particularly the alloys with high silicon content, requires the use of tools with acute rake and clearance angles. In general, rake angles are increased as silicon content is decreased. The same tool geometry and cutting tools used to turn high-silicon aluminum alloys such as 390 apply to boring operations of this same alloy (see the section "Turning of High-Silicon Aluminum Alloys" in this article).

Tool Material. Although high-speed steel tools are used to some extent for boring aluminum, carbide tools permit higher surface speeds (up to 300 m/min, or 1000 sfm, or more), with markedly longer tool life.

Table 16 Maximum depths per drill entry for drilling in automatic bar and chucking machines

	Depth, in drill diameters						
	First drill			Second drill			
Drill type	First entry	Second entry	Third entry	First entry	Second entry	Third entry	Subsequent entries
Standard twist drill	4	1½	¾	...	...	...	...
High or low helix, or straight flute	5	2	1	1½	¾	½	½

Table 17 Feeds for drilling aluminum alloys in automatic bar and chucking machines

Drill diameter		Tolerance		Feed			
				2011-T3		Other alloys	
mm	in.	mm	in.	mm	in.	mm	in.
1.59	0.0625	±0.038	±0.0015	0.102	0.0040	0.10	0.004
3.18	0.125	±0.05	±0.002	0.305	0.0120	0.25	0.010
4.75	0.187	±0.05	±0.002	0.366	0.0144	0.30	0.012
6.35	0.250	±0.05	±0.002	0.426	0.0168	0.36	0.014
9.53	0.375	±0.064	±0.0025	0.518	0.0204	0.43	0.017
12.70	0.500	±0.064	±0.0025	0.518	0.0204	0.43	0.017
19.05	0.750	±0.08	±0.003	0.518	0.0204	0.43	0.017

Table 18 Relation of drill diameter to approximate maximum depth of hole in gun drilling

Drill diameter		Depth of hole, number of drill diameters
mm	in.	
≤13	≤½	12
13–25	½–1	7
25–38	1–1½	6
38–50	1½–2	5
50–64	2–2½	4
64–75	2½–3	3

Carbide cutters readily yield surface finishes of 0.25 to 0.50 μm (10 to 20 μin.).

When used in conjuction with precision boring machines, diamond tools can produce surface finishes of 0.025 μm (1 μin.). They are also capable of holding size over a long production run. If the cut is continuous and the work metal contains no hard spots, diamond tools are most effective in boring the abrasive high-silicon alloys.

Tool Sharpening. For optimum tool life, cutting edges and adjacent surfaces must be free of burrs and scratches. The hand stoning of cutting edges with an oil stone is recommended. When a carbide boring tool is sharpened with a 400- or 500-grit diamond wheel, surface finishes of 0.075 to 0.10 μm (3 to 4 μin.) can be obtained.

Speed and Feed. The optimum speed for boring aluminum alloys depends somewhat on alloy and temper, but largely on tool material and whether the operation is roughing or finishing. Selection of feed depends largely on tool material and whether the operation is roughing or finishing.

Nominal speeds and feeds for boring with high-speed steel and carbide tools are given in Table 9; speeds and feeds for boring aluminum alloys with polycrystalline diamond tools are given in Table 10. Speed is increased for a shallower depth of cut and lighter feed, regardless of the alloy or tool material.

Depth of Cut. The speeds and feeds given in Tables 9 and 10 are based on a 2.5 mm (0.10 in.) depth of cut for roughing and 0.25 mm (0.010 in.) for finishing. Depth of cut is sometimes greater than 2.5 mm (0.10 in.) for rough boring if power is available and the setup can be made sufficiently rigid. Finishing cuts significantly less than 0.25 mm (0.010 in.) in depth are seldom used.

Cutting fluid is recommended, but boring has been done dry. There is usually some sacrifice of productivity, dimensional accuracy, or surface finish when aluminum is bored without a cutting fluid. A mixture of one part soluble oil to 20 to 30 parts water is most commonly used. In addition, mineral oil, or mineral oil mixed with up to 50% lard oil, is often used, especially when the best possible surface finish is desired. Mixtures of kerosenes and lubricating oil are sometimes used.

Procedures and equipment common to the boring of all metals are covered in detail in the article "Boring" in this Volume. Specific procedures for boring an aluminum alloy are described in the example that follows.

Example 2: The Boring of Concentric Holes. Two small and two large holes were rough and finish bored from opposite sides of a heat-treated aluminum forging, as shown in Fig. 15. It was required that the large and small holes be concentric within 0.010 mm (0.0004 in.) total indicator reading (TIR), parallel within 0.008 mm (0.0003 in.), and square with the surface within 0.008 mm (0.0003 in.). The operation was done in a double-end precision boring mill, and carbide tools were used. Details are given with Fig. 15.

Planing and Shaping

The techniques and equipment used for the planing and shaping of other metals are generally applicable to aluminum alloys. Planing methods are discussed in the article "Planing" in this Volume. Similar information for shaping appears in the article "Shaping and Slotting" in this Volume.

Planing. Either high-speed or carbide tools for planing aluminum usually have a back rake angle of 30° or more (sometimes as much as 60°) and a 0° side rake. A speed of 90 m/min (300 sfm) is generally recommended as maximum; however, this speed is higher than can be obtained with most planers.

Roughing feeds are about 2.3 mm (0.090 in.) per stroke when depth of cut is near the recommended maximum of 13 mm (½ in.). When shallow cuts are necessary (for lack of power or other reasons), a heavier feed can be used. For a 2.5 mm (0.10 in.) depth of cut, feed can be increased to about 3.18 mm (0.125 in.) per stroke. The finish planing of aluminum alloys is usually done with a cut no deeper than 0.25 to 0.38 mm (0.010 to 0.015 in.) and a feed equal to three-fourths the width of the broad-nose finishing tool.

Aluminum alloys are often planed without a cutting fluid. A cutting fluid is helpful in producing a better surface finish, but flooding with cutting fluid is seldom feasible. When surface finish is a primary objective, application of a mixture of kerosene and lubricating oil, or of kerosene and lard oil, by means of a swab is common practice. Sprayed cutting fluid is effective. Soluble-oil mixtures are ordinarily used in cutting fluid spray systems. Table 11 lists nominal speeds and feeds for the planing of wrought and cast aluminum.

Shaping. Tool forms, tool materials, and cutting fluids for shaping aluminum are the same as those for planing. Maximum speed in shaping is usually the maximum ram speed of the machine. Feed and depth of cut also depend to some extent on machine capabilities. A feed of 0.51 to 0.76 mm/stroke (0.020 to 0.030 in./stroke) for roughing and about 0.25 mm/stroke (0.010 in./stroke) for finishing are common. Depth of cut is often as much as 0.25 mm (0.10 in.) for roughing and 0.51 mm (0.020 in.) or less for finishing.

Broaching

All aluminum alloys can be broached successfully with standard broaching equipment and with the same general procedures as for other metals. However, better surface finish and dimensional accuracy are obtained with heat-treated alloys. Details of equipment and procedures that are common to the broaching of all metals are covered in the article "Broaching" in this Volume. The principal limitation in broaching aluminum is the difficulty in maintaining an accurate relation between the broached hole and

Table 19 Nominal speeds and feeds for the gun drilling of aluminum alloys with carbide tools

Material	Hardness, HB (500 kg)	Condition	Speed, m/min (sfm)	Feed, mm/rev (in./rev)(a), at a nominal hole diameter of: 2–4 mm (0.078–0.156 in.)	4–6 mm (0.156–0.250 in.)	6–12 mm (¼–½ in.)	12–18 mm (½–¾ in.)	18–25 mm (¾–1 in.)	25–50 mm (1–2 in.)	Carbide tool material grade, ISO (C)
Wrought										
EC 3005 6066, 1060 4032 6070, 1100 5005 6101, 1145 5050 6151, 1175 5052 6253, 1235 5056 6262, 2011 5083 6463, 2014 5086 6951, 2017 5154 7001, 2018 5252 7004, 2021 5254 7005, 2024 5454 7039, 2025 5456 7049, 2117 5457 7050, 2218 5652 7075, 2219 5657 7079, 2618 6053 7175, 3003 6061 7178, 3004 6063	30–80	Cold drawn	215	0.004–0.006	0.008–0.013	0.025–0.063	0.038–0.075	0.050–0.102	0.075–0.13	K20
			(700)	(0.00015–0.00025)	(0.0003–0.0005)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(0.003–0.005)	(C-2)
	75–150	Solution treated and aged	200	0.004–0.006	0.008–0.013	0.025–0.063	0.038–0.075	0.050–0.102	0.075–0.13	K20
			(650)	(0.00015–0.00025)	(0.0003–0.0005)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(0.003–0.005)	(C-2)
Cast										
Sand and permanent mold										
A140 F332.0 520.0, 201.0 333.0 535.0, 208.0 354.0 705.0, 213.0 355.0 707.0, 222.0 C355.0 A712.0, 224.0 356.0 D712.0, 242.0 A356.0 713.0, 295.0 357.0 771.0, B295.0 359.0 850.0, 308.0 B443.0 A850.0, 319.0 514.0 B850.0, 328.0 A514.0, A332.0 B514.0, Hiduminium RR-350	40–100	As-cast	200	0.004–0.006	0.008–0.013	0.025–0.063	0.038–0.075	0.050–0.102	0.075–0.13	K20
			(650)	(0.00015–0.00025)	(0.0003–0.0005)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(0.003–0.005)	(C-2)
	70–125	Solution treated and aged	200	0.004–0.006	0.008–0.013	0.025–0.063	0.038–0.075	0.050–0.102	0.075–0.13	K20
			(650)	(0.00015–0.00025)	(0.0003–0.0005)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(0.003–0.005)	(C-2)
Die castings										
360.0 A380.0, A360.0 C443.0, 380.0 518.0	40–100	As-cast	200	0.004–0.006	0.008–0.013	0.025–0.063	0.038–0.075	0.050–0.102	0.075–0.13	K20
			(650)	(0.00015–0.00025)	(0.0003–0.0005)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(0.003–0.005)	(C-2)
	70–125	Solution treated and aged	200	0.004–0.006	0.008–0.013	0.025–0.063	0.038–0.075	0.050–0.102	0.075–0.13	K20
			(650)	(0.00015–0.00025)	(0.0003–0.0005)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(0.003–0.005)	(C-2)
383.0, A384.0, 413.0, A413.0	40–100	As-cast	200	0.004–0.006	0.008–0.013	0.025–0.063	0.038–0.075	0.050–0.102	0.075–0.13	K20
			(650)	(0.00015–0.00025)	(0.0003–0.0005)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(0.003–0.005)	(C-2)
	70–125	Solution treated and aged	200	0.004–0.006	0.008–0.013	0.025–0.063	0.038–0.075	0.050–0.102	0.075–0.13	K20
			(650)	(0.00015–0.00025)	(0.0003–0.0005)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(0.003–0.005)	(C-2)
390.0, 392.0	40–100	As-cast	150	0.004–0.006	0.008–0.013	0.020–0.050	0.025–0.063	0.038–0.075	0.050–0.102	K20
			(500)	(0.00015–0.00025)	(0.0003–0.0005)	(0.0008–0.002)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(C-2)
	70–125	Solution treated and aged	135	0.004–0.006	0.008–0.013	0.020–0.050	0.025–0.063	0.038–0.075	0.050–0.102	K20
			(450)	(0.00015–0.00025)	(0.0003–0.0005)	(0.0008–0.002)	(0.001–0.0025)	(0.0015–0.003)	(0.002–0.004)	(C-2)

(a) Feeds are specified for single-flute gun drills. Source: Ref 5

other surfaces of the workpiece, even when the starting hole is accurately located.

Tool Material. The general-purpose high-speed steels, such as M2, are used for most tools for broaching aluminum. In some high-production operations, especially when broaching high-silicon alloys, broaches made of the more highly alloyed high-speed steels or of carbide have proved economical. Surface treatments such as chromium plating or oxidizing help to prolong the life of high-speed steel broaches.

A fine finish on the tool can be important. In one application, the life of a high-speed steel broach was increased from 2000 to 7400 pieces when the cutting edges were wet blasted with a superfine abrasive.

Broach Design. In rough broaching, a coarse tooth pitch is desirable, with only two or three teeth in contact and cutting at any one time. For internal finish broaching, the best results are obtained if only two teeth are cutting; in external finish broaching, it is often best to have only one tooth engaging at a time.

Broaches used for aluminum should have a face angle of 10°. For external surface broaching, the clearance angle should be 3½°; for internal broaching, 2°. Although a large clearance angle provides better cutting action, it markedly reduces broach life. Therefore, clearance angles should be kept to a minimum to reduce loss of size when the broach is sharpened.

Speed and Feed. A speed range of 9.1 to 15 m/min (30 to 50 sfm) is generally recommended for broaching aluminum alloys. When rigidity, supply of cutting fluid, and hardness of the work metal are nearly ideal, the upper portion of the range can be used. When one or more of these conditions is less than ideal, the speed can be reduced.

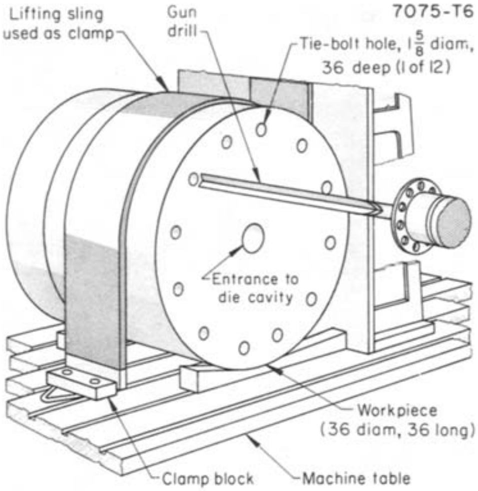


Fig. 19 Setup for deep-hole drilling an explosive forming die in a boring mill. Dimensions given in inches

Greater feed per tooth (chip load) is recommended for broaching aluminum alloys than for steel. A feed of 0.15 mm/tooth (0.006 in./tooth) is usually optimum for spline broaching and about 0.08 mm/tooth (0.003 in./tooth) for broaching round holes. Table 12 lists nominal speeds and feeds for machining both wrought and cast aluminum alloys.

Cutting fluid is required for best results. A copious supply of soluble oil mixed with water is satisfactory for most applications. However, mineral oil, or mineral oil mixed with lard oil, will usually improve surface finish (see the section "Cutting Fluids" in this article).

Production Practice. The following example provides details of typical practice in broaching aluminum. For the workpiece described in this example, broaching was the most feasible and least costly method of obtaining the required results.

Example 3: The Broaching of Internal Splines. Internal splines were broached in a 2017-T4 aluminum clutch housing (Fig. 16). The splined hole contained eight teeth with 32 diametral pitch, 20° pressure angle, and partial dedendum; the internal diameter was 5.550 mm (0.2185 in.). The length of the broaching cut was 7.49 mm (0.295 in.). The pull broach (details in Fig. 16) was made of high-speed steel, and broaching was done in a 9 kN (1 tonf) vertical machine. Processing data are given in the table accompanying Fig. 18.

Broaching of 390 High-Silicon Aluminum Alloy (Ref 2). Horizontal broaching has long been the preferred method for rough and finish machining for such complex cast iron castings as cylinder heads and engine blocks. Broaching permits several surfaces on such parts to be machined in exact relationship to each other because this relationship can be build into the broach tooling. Broaching also permits very high stock removal rates, and machine downtime is typically low.

Because of their low hardness, modulus, and strength, aluminum casting alloys have traditionally been considered unlikely candidates for this type of broaching. The cutting speed capability typical of a broach designed to machine cast iron is slow (7.6 to 46 m/min, or 25 to 150 sfm). Most aluminum alloys are gummy at such speeds; because of relatively low hardness, chips clog the cutting teeth of the broach, and the surface of the resulting broached casting is torn and rough. The unit tool forces, characteristic of such a broach, result in breakage of thinner casting sections and edge breakout on other sections (chipping of the trailing edge of the workpiece as the tool exits). The low modulus of most aluminum castings, when compared to that of iron, also results in additional fixturing and clamping requirements.

One study concluded that alloy 390 could be broached at cutting speeds and tool geometries typical of existing iron cutting broaches with carbide tooling. Tool life was much better than that achieved for cast iron (Fig. 17). It also becomes evident that surface finishes better than 2.50 mm (100 μin.) could be obtained as long as the uncut chip thickness of each cutting edge was kept below 0.10 μm (0.004 in.) (Fig. 18). Such chip loads are typical of finishing cuts when broaching cast iron, and the surface finishes obtained are comparable to or better than those obtained with cast iron. The use of coolant had little effect on tool life or surface finish; however, coolant would tend to aid in chip disposal.

Tests have shown that varying the rake angles, shear angles, and chip loads can be effective in reducing cutting forces during broaching. This approach to low-force tooling may not be needed for alloy 390, because the cutting forces are less than one-half the forces for cast iron in comparable roughing cuts using negative-rake tooling. However, these concepts have proved successful in reducing breakage while broaching lighter-weight, thin-wall gray iron parts and will also perform successfully when broaching alloy 390.

Drilling in Drill Presses

Although the standard twist drills and drilling equipment used for steel can be employed in drilling aluminum alloys, optimum results require drills of special design as well as higher rotational speeds and heavier feeds. Drills for aluminum are usually made with deep, well-polished flutes, narrow margins, and large helix angles. Proper drill design and drilling practice will frequently permit the removal of three to four times as much aluminum as steel per unit of power. The article "Drilling" in this Volume discusses general equipment and practice.

Table 20 Reamer design and operating conditions for reaming aluminum alloys

Reamer design	
Reamer size	
Hand reaming, number times drill diameter	1.01
Machine reaming, number times drill diameter	1.02
Flute type	Straight to 10° spiral
Tooth spacing	
Straight flute	Uneven(a)
Spiral flute	Even
Tooth style	
Roughing	Solid or nicked
Finishing	Solid
Top rake, degrees	5–8
Clearance angles, degrees	
Primary	4–7
Secondary	15–20
Cutting angle, degrees	85–91
Land width, mm (in.)	0.51–1.52 (0.020–0.060)(b)
Operating conditions	
Speed, roughing, m/min (sfm)	
Hard alloys	≤60 (200)
Soft alloys	18–30 (60–100)
Speed, finishing, m/min (sfm)	
Straight reamers	≤120 (400)
Taper reamers	≤90 (300)
Feed, mm/rev (in./rev)	
Roughing	0.33–0.89 (0.013–0.035)
Finishing	0.08–0.25 (0.003–0.010)

(a) To avoid chatter. (b) Land width is approximately 1/25 of reamer diameter.

For drilling in drill presses, the helix angle of the drill should be increased with the depth of the hole to be drilled, ranging from a low-helix 24° angle for very shallow holes in thin stock to a high-helix (40 to 48°) angle for deep holes, for which freer cutting is important (Table 13). The high-helix drill has a more acute cutting angle, resulting in more rapid penetration and freer and cleaner cutting. Lands and margin are narrower than on the low-helix drill, resulting in reduced friction and increased chip space in the flutes. The slightly greater resistance to chip movement can be overcome by polishing the flutes and supplying ample cutting fluid. This type of drill is recommended for deep holes, but is unsuitable for drilling thin stock because of its tendency to hog in.

A twist drill with a 28° helix is suitable for holes of medium depth up to about six drill diameters. A drill with a 24° (or less) helix is recommended for thin stock because it has less tendency to overfeed.

The point angle supplied on standard twist drills is 116 to 118°. The angle should be about 130 to 140° for drilling most aluminum alloys to facilitate chip removal and to minimize burring. However, drills for high-silicon alloys should have a less obtuse point, down to about 90°, for ease of penetration. For drilling thin sheet, the point angle should be very obtuse to permit the drill to cut to its full diameter before the point breaks through. With this type of drill,

Table 21 Nominal speeds and feeds for the reaming of aluminum alloys with high-speed tool steel and carbide tools

Material	Hardness, HB (500 kg)	Condition	Roughing: Speed, m/min (sfm)	Roughing: Feed, mm/rev (in./rev)(a), at a reamer diameter, mm (in.), of: 3 (1/8)	6 (1/4)	12 (1/2)	25 (1)	35 (1.5)	50 (2)	Roughing: Tool material grade, ISO (AISI or C)	Finishing: Speed, m/min (sfm)	Finishing: Feed, mm/rev (in./rev)(a), at a reamer diameter, mm (in.), of: 3 (1/8)	6 (1/4)	12 (1/2)	25 (1)	35 (1.5)	50 (2)	Finishing: Tool material grade, ISO (AISI or C)
Wrought																		
EC, 1060, 1100, 1145, 1175, 1235, 2011, 2014, 2017, 2018, 2021, 2024, 2025, 2117, 2218, 2219, 2618, 3003, 3004, 3005, 4032, 5005, 5050, 5052, 5056, 5083, 5086, 5154, 5252, 5254, 5454, 5456, 5457, 5652, 5657, 6053, 6061, 6063, 6066, 6070, 6101, 6151, 6253, 6262, 6463, 6951, 7001, 7004, 7005, 7039, 7049, 7050, 7075, 7079, 7175, 7178	30–80	Cold drawn	150 (500)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	55 (180)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	S3, S4, S2 (M1, M2, M7)
			305 (1000)	0.13 (0.005)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)	76 (250)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	K20 (C-2)
	75–150	Solution treated and aged	150 (500)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	55 (180)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	S3, S4, S2 (M1, M2, M7)
			305 (1000)	0.13 (0.005)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)	76 (250)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	K20 (C-2)
Cast																		
Sand and permanent mold																		
A140, 201.0, 208.0, 213.0, 222.0, 224.0, 242.0, 295.0, B295.0, 308.0, 319.0, 328.0, A332.0, F332.0, 333.0, 354.0, 355.0, C335.0, 356.0, A356.0, 357.0, 359.0, B443.0, 514.0, A514.0, B514.0, 520.0, 535.0, 705.0, 707.0, A712.0, D712.0, 713.0, 771.0, 850.0, A850.0, B850.0, Hiduminium RR-350	40–100	As-cast	150 (500)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	49 (160)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	S3, S4, S2 (M1, M2, M7)
			305 (1000)	0.13 (0.005)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)	76 (250)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	K20 (C-2)
	70–125	Solution treated and aged	120 (400)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	49 (160)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	S3, S4, S2 (M1, M2, M7)
			260 (850)	0.13 (0.005)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)	76 (250)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	K20 (C-2)
Die castings																		
360.0, A360.0, 380.0, A380.0, C443.0, 518.0	40–100	As-cast	150 (500)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	37 (120)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	S3, S4, S2 (M1, M2, M7)
			305 (1000)	0.13 (0.005)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)	60 (200)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	K20 (C-2)
	70–125	Solution treated and aged	120 (400)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	37 (120)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	S3, S4, S2 (M1, M2, M7)
			260 (850)	0.13 (0.005)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)	60 (200)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	K20 (C-2)
383.0, A384.0, 413.0, A413.0	40–100	As-cast	105 (350)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	30 (100)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	S3, S4, S2 (M1, M2, M7)
			215 (700)	0.13 (0.005)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)	53 (175)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	K20 (C-2)
	70–125	Solution treated and aged	105 (350)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	30 (100)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	S3, S4, S2 (M1, M2, M7)
			215 (700)	0.13 (0.005)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)	53 (175)	0.102 (0.004)	0.15 (0.006)	0.25 (0.010)	0.40 (0.015)	0.45 (0.018)	0.50 (0.020)	K20 (C-2)
390.0, 392.0	40–100	As-cast	30 (100)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	24 (80)	0.13 (0.005)	0.20 (0.008)	0.30 (0.012)	0.45 (0.018)	0.50 (0.020)	0.65 (0.025)	S3, S4, S2 (M1, M2, M7)
			60 (200)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	K20 (C-2)	30 (100)	0.13 (0.005)	0.20 (0.008)	0.30 (0.012)	0.45 (0.018)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)
	70–125	Solution treated and aged	30 (100)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	S3, S4, S2 (M1, M2, M7)	24 (80)	0.13 (0.005)	0.20 (0.008)	0.30 (0.012)	0.45 (0.018)	0.50 (0.020)	0.65 (0.025)	S3, S4, S2 (M1, M2, M7)
			60 (200)	0.18 (0.007)	0.25 (0.010)	0.40 (0.015)	0.50 (0.020)	0.65 (0.025)	0.75 (0.030)	K20 (C-2)	30 (100)	0.13 (0.005)	0.20 (0.008)	0.30 (0.012)	0.45 (0.018)	0.50 (0.020)	0.65 (0.025)	K20 (C-2)

(a) Based on four flutes for 3 and 6 mm (1/8 and 1/4 in.) reamers, six flutes for 12 mm (1/2 in.) reamers, and eight flutes for 25 mm (1 in.) and larger reamers. Source: Ref 5

a spur point may be necessary to assist in centering.

The standard lip clearance of 12 to 13° should be increased to about 17° for heavy feeds and for softer alloys. Insufficient lip clearance will cause excessive drill breakage. The drill cutting lips must be keen and smooth, and all surfaces over which the chip passes must be polished to minimize friction and chip buildup. Recommended point angles, helix angles, and lip clearances are summarized in Table 13.

Drill Material. Most drills for aluminum are made of high-speed tool steels; M1, M7, and M10 are the most common grades. Only rarely can the additional cost for drills made of a more highly alloyed grade of high-speed tool steel or of carbide be justified for conventional drilling in drill presses.

Speed and Feed. With most drill presses, the peripheral speed of small-diameter drills is relatively low; therefore, such drills can be operated at the maximum efficient rotational speed of the machine. In general, high-speed tool steel drills can be operated at a maximum of about 180 m/min (600 sfm). When variable speed is available, drill life can be increased in drilling deeper holes by bringing the drill up to speed gradually.

Because of the ease of penetrating most aluminum alloys, feeds up to twice those used for drilling steel can be employed. Feed varies with drill diameter; the larger-diameter drills permit heavier feeds, as indicated below:

Feed, mm/rev	Feed, in./rev	Drill diameter, mm	Drill diameter, in.
0.025	0.001	1.6	1/16
0.08	0.003	3.2	1/8
0.18	0.007	6.4	1/4
0.30	0.012	13	1/2
0.41	0.016	19	3/4
0.48	0.019	25	1
0.64	0.025	38	1½
0.76	0.030	50	2

Nominal speeds and feeds for wrought and cast aluminum alloys are listed in Table 14. Table 15 provides speed and feed data using spade drills.

Cutting Fluid. Drilling of thin sections does not require a cutting fluid, but it is essential to both drill life and hole quality that a copious supply of cutting fluid be provided for all deep-hole drilling. Soluble-oil emulsions or kerosene and lard oil mixtures are satisfactory for general drilling. In drilling to a depth greater than six times the drill diameter, the workpiece should be kept cool by spraying. In addition, the drill should be withdrawn several times during drilling to ensure that the cutting fluid floods the hole completely.

Drilling Operations in Automatic Bar and Chucking Machines

When drilling is done in multiple-operation machines, it is especially important that the tool be correctly ground and set (see the section "Automatic Lathes" in the article "Multiple-Operation Machining" in this Volume). Standard twist drills are generally used for drilling holes up to six diameters deep. For drilling deeper holes, drills that pass the chips up the flutes more readily are recommended. When machining 2017-T4, 2024-T4, and 2011-T3, straight-flute drills are generally used for drilling holes deeper than six diameters. Either high-helix or low-helix drills can be used for drilling deep holes. The web of the high-helix drill is uniform in thickness along the entire length of the body, providing large flutes for the chips. Wide flutes also characterize the low-helix drill. Both types give good results, particularly in the drilling of soft or gummy alloys. For holes with a high ratio of depth to diameter, half-round or gun drills are sometimes used.

When standard twist drills are used for drilling deeper than three diameters, it may be necessary to enlarge the flutes. This can be done with a thin grinding wheel that has been dressed to a radius. The flute is held at an angle to the wheel to ensure the proper curvature in the flute. All flutes on a drill should be ground alike.

Drill Margin. Margins along the edges of flutes support the drill in the hole and keep the drill cutting the correct diameter. When drilling aluminum alloys, standard drill margins can often be reduced in width without loss of necessary support. The narrower margin reduces friction between the drill and the hole, thus reducing the amount of heat generated. In many production jobs, a narrower margin greatly increases drill life.

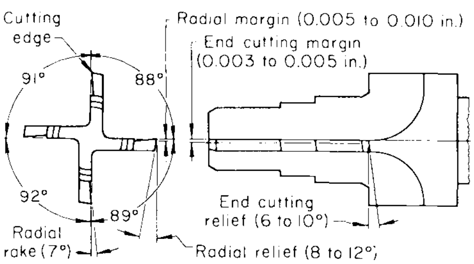


Fig. 20 Design of a finishing reamer used for reaming aluminum in automatic bar and chucking machines. Dimensions given in inches

Table 22 Nominal speeds for the tapping of aluminum alloys

Alloy	Speed, m/min	Speed, sfm
Nonheat-treated cast alloys	35	115
Heat-treated cast alloys, solution treated and aged	27	90
Cold-drawn wrought alloys	38	125
Heat-treated wrought alloys, solution treated and aged	30	100

Web Thickness. For drilling aluminum alloys, web thickness at the point should be reduced as the drill is ground back. This will reduce the end pressure on the drill because the chisel point does not cut but compresses the metal ahead of it. Generally, the web can be somewhat thinner at the point without making the drill susceptible to breakage

Table 23 Nominal speeds for the tapping of aluminum alloys with high-speed tool steels

Material				Hardness, HB (500 kg)	Condition	Speed, m/min (sfm) at a pitch, mm (threads/in.), of: >3 (≤7)	1.5–3 (8–15)	1–1.5 (16–24)	≤1 (>24)	High-speed steel tool material grade, ISO (AISI)
Wrought										
EC	2218	5252	6253	30–80	Cold drawn	17	29	37	38	S2, S3
1060	2219	5254	6262			(55)	(95)	(120)	(125)	(M10, M7, M1)
1100	2618	5454	6463	75–150	Solution treated	14	23	29	30	S2, S3
1145	3003	5456	6951		and aged	(45)	(75)	(95)	(100)	(M10, M7, M1)
1175	3004	5457	7001							
1235	3005	5652	7004							
2011	4032	5657	7005							
2014	5005	6053	7039							
2017	5050	6061	7049							
2018	5052	6063	7050							
2021	5056	6066	7075							
2024	5083	6070	7079							
2025	5086	6101	7175							
2117	5154	6151	7178							
Cast										
Sand and permanent mold										
A140	319.0	357.0	A712.0	40–100	As-cast	15	26	34	35	S2, S3
201.0	328.0	359.0	D712.0			(50)	(85)	(110)	(115)	(M10, M7, M1)
208.0	A332.0	B443.0	713.0	70–125	Solution treated	12	20	26	27	S2, S3
213.0	F332.0	514.0	771.0		and aged	(40)	(65)	(85)	(90)	(M10, M7, M1)
222.0	333.0	A514.0	850.0							
224.0	354.0	B514.0	A850.0							
242.0	355.0	520.0	B850.0							
295.0	C355.0	535.0								
B295.0	356.0	705.0								
308.0	A356.0	707.0								
Hiduminium RR-350										
Die castings										
360.0	A380.0			40–100	As-cast	20	34	43	46	S2, S3
A360.0	C443.0					(65)	(110)	(140)	(150)	(M10, M7, M1)
380.0	518.0			70–125	Solution treated	14	23	29	30	S2, S3
					and aged	(45)	(75)	(95)	(100)	(M10, M7, M1)
383.0				40–100	As-cast	20	34	43	46	S2, S3
A384.0						(65)	(110)	(140)	(150)	(M10, M7, M1)
413.0				70–125	Solution treated	14	23	29	30	S2, S3
A413.0					and aged	(45)	(75)	(95)	(100)	(M10, M7, M1)
390.0				40–100	As-cast	17	30	34	37	S2, S3
392.0						(55)	(100)	(110)	(120)	(M10, M7, M1)
				70–125	Solution treated	12	21	27	29	S2, S3
					and aged	(40)	(70)	(90)	(95)	(M10, M7, M1)

(a) These speeds are for tapping 65 to 75% threads in shallow through holes. Reduce the speed when tapping deep holes, blind holes, or a higher percentage of thread. Source: Ref 5

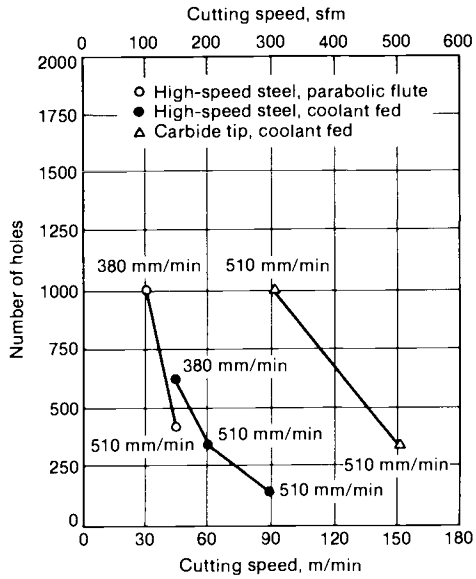
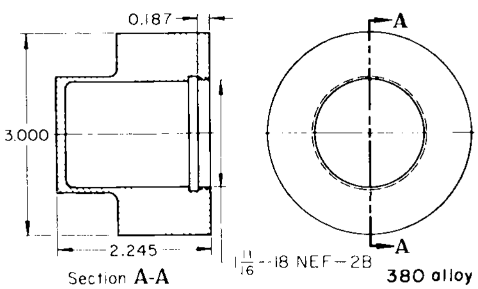


Fig. 21 Drill life at various penetration rates for several types of drills when drilling permanent mold 390 aluminum alloy. Source: Ref 2



Details of single-point threading tool

Material	Carbide
Cutting edge angle, degrees	60
Top rake angle, degrees	20
Clearance angle, degrees	7

Operating conditions

Speed, m/min (sfm)	145 (475)
Diameter of bore, mm (in.)	43 ($1^{11}/_{16}$)
Length of threads, mm (in.)	4.75 (0.187)
Cutting fluid	Soluble oil:water (1:25)
Setup time, h	1
Cycle time, s	40
Production rate, pieces/h	75

Fig. 22 Die casting threaded in a single-spindle chucking machine with a threading attachment. Dimensions in figure given in inches

Table 24 Geometry and cutting conditions for the drilling and tapping of alloy 390

Type	Speed m/min	Speed sfm	Feed	Coolant
Drills				
Tungsten carbide tipped, coolant fed	90	300	508 mm/min (20 in./min)	34 L/min (9 gal./min) at 3.5 MPa (500 psi), soluble oil (20:1)
High-speed steel, high helix, parabolic flute, noncoolant fed	20	70	381 mm/min (15 in./min) at 170 kPa (25 psi)	19 L/min (5 gal./min), soluble oil (20:1)
High-speed steel, regular helix, coolant fed	20	70	4.6 m/min (15 sfm)	34 L/min (9 gal./min) at 3.5 MPa (500 psi), soluble oil (20:1)
Taps				
Thread forming, high-speed steel, four lubricant grooves	20	70	Determined by pitch	19 L/min (5 gal./min), soluble oil (20:1)

Source: Ref 2

when drilling aluminum alloys than when drilling steel.

For small-diameter drills, notched-point thinning is common. On larger drills, this type of point may produce a poor chip; therefore, the entire flute is ground at the point. The notched point is obtained by using the sharp corner of an abrasive wheel with the side of the wheel following the angle of the chisel point. The drill should be held at an angle to the wheel to form a slight rake for the new cutting edge, which is ground to the center of the point of the drill.

When the entire flute at the point is thinned by grinding, a thin wheel that has been dressed to a radius is used. The drill is held so that the flute is at an angle to the wheel, as when widening the flute. Most of the metal should be ground off the back of the land, and care must be taken not to grind the rake formed by the helix angle from the cutting edge or to destroy the shape of the cutting edge. Thinning of the web must be done uniformly in each flute to ensure that the cut remains balanced on both sides of the centerline of the drill.

Cutting-Lip Angle. In general drilling practice, if holes are not too deep, a standard included cutting-lip angle of 118° will give a satisfactory performance. However, if deeper holes are drilled with this cutting-lip angle, the chip produced does not come out easily. For drilling deeper holes, larger included cutting-lip angles are used, forming a narrower chip that readily passes up the flutes.

Occasionally, smaller cutting-lip angles can be used because the transverse forces at the cutting edges are greater and the tool can drill without runout. However, a broader chip is produced. When several drills are used in the same hole, each succeeding tool should have a slightly more blunt cutting-lip angle than that of the preceding one, so that it can center at the outside of the cutting edges.

The clearance angle behind the cutting edges should be 12 to 20°. Larger clearances are used for straight-flute drills and for drills ground with large cutting-lip angles. Clearance should extend from periphery to center so that the chisel point is at an angle of 130 to 145° with the cutting edges.

Rake angles are set by the helix angle of the drill. For standard twist drills, this is usually 20 to 25°; for high-helix drills, 40 to 43°; for low-helix drills, 7 to 15°; and for straight-flute and half-round drills, 0°.

Center Drills. When a drill starts against a flat surface, it may skid sideways before cutting, particularly if it is small or protrudes considerably from the holder. This may cause the drill to break or to cut off-center. Therefore, except with large, rigidly held drills, a center drill is recommended to provide an accurate start.

For maximum rigidity, center drills should be of relatively large diameter and short length. Unless a countersink of a special angle is to be left in the workpiece, an included cutting-lip angle of 90° should be used. In general, for proper centering, the outside of the cutting lips of the drill that follows the center drill should strike the stock first, ensuring adequate support.

Depth of Hole. In multiple-operation machines, to prevent chips from jamming in the flutes and causing drill breakage when deep holes are being drilled, a limit must be set on the depth drilled with each entry of the tool. The maximum depths, in terms of drill diameter, that can be drilled per entry under normal production conditions are given in Table 16. Before reentry, the drill should be completely backed out of the hole so that the chips can be washed away.

The type of workpiece, the number of positions on the machine available for drilling, and the type of machine determine the practice for drilling deep holes. It is sometimes economical to use several drills for drilling deep holes.

For holes more than eight diameters deep, half-round or gun drills are often used. To ensure adequate support for the half-round drill when it starts cutting, another type of drill should be used to drill a starting hole three diameters or more deep. The half-round drill can drill four diameters deeper in the first entry of the started hole.

Drill Speed. Generally, small-diameter, high-speed tool steel drills can be operated at speeds up to 180 m/min (600 sfm). However, when larger drills are used to remove large quantities of metal at a rapid rate,

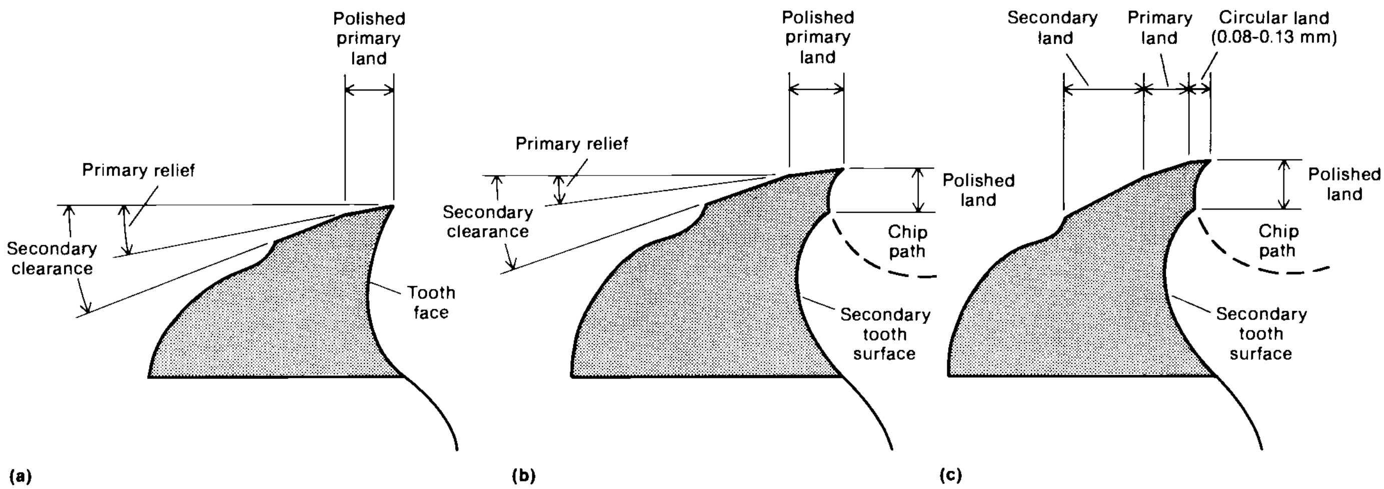


Fig. 23 Three end mill geometries designed for the cutting of aluminum alloys. (a) Conventional milling. (b) Heavy metal removal. (c) Finishing operations. Source: Ref 1

lower speeds will give more economical drill life. A guide for choosing proper speeds for most conditions is:

Drill diameter, mm	Drill diameter, in.	Speed, m/min	Speed, sfm
<25	<1	180	600
25–38	1–1½	170	550
>38	>1½	140	450

Feed for drilling in multiple-operation machines depends on the size and strength of the drill; finish, tolerance, and concentricity desired; and power available. As feed is increased, torque on the drill increases, until the breaking point may be reached. On the other hand, lower feed produces thinner chips, which are more likely to clog the flutes than thicker chips. Clogging is likely to break the drill or mar the finish of the workpiece.

Recommended drill feeds are given in Table 17 for drill sizes up to 19.05 mm (0.750 in.) in diameter. Feeds for larger drills usually depend on the power available in the machine.

For greater accuracy and better finish, lower feed may be necessary. For deep holes, machined with several entries, feed should be decreased 15% for each successive entry. Lower feed should be used to drill thin-wall parts.

Power Requirements. When large quantities of metal are removed, lower feed may be necessary to keep the power within the limits available in the machine. When drilling with standard drills (ground to 118° included cutting-lip angle and 15° clearance) to four diameters deep in 2017-T4 and 2024-T4, the metal removal rate is 18 000 to 24 000 mm^3/min/kW (1.5 to 2 $in.^3$/min/hp). For 2011-T3, the rate is 31 000 to 37 000 mm^3/min/kW (2.5 to 3.0 $in.^3$/min/hp). These figures are based on a feed of 0.43 mm/rev (0.017 in./min). With lower feed, the rate of metal removal decreases.

Drill Size Versus Hole Size. Drills, when properly ground and set, will cut aluminum alloys to size or not more than 0.05 mm (0.002 in.) oversize. However, any condition that causes overheating is likely to decrease the size of the hole when it is measured after the workpiece has cooled. This is especially noticeable when drilling large holes. The drilling of deep holes at a high feed rate will decrease hole size in relation to drill size. Other tools working simultaneously will sometimes generate enough heat to cause the production of holes close to or even smaller than drill size, as measured after the workpiece has cooled to room temperature.

Deep-Hole Drilling

Gun drills, tipped with either carbide or high-speed tool steel, have replaced twist drills for many deep-hole drilling applications, regardless of the metal being drilled. In gun drilling, the maximum depth of hole that can be drilled successfully is generally related to drill size (Table 18). The length-to-diameter relationships shown in Table 18 are generally standard, although gun drills having much greater length-to-diameter ratios are often used. Table 19 lists nominal speeds and feeds for the gun drilling of wrought and cast aluminum alloys.

Gun drills, with a single-flute cutting head and grooved shank, must have a sufficient flow of cutting fluid under high pressure at the point where the cutting edge contacts the work; this keeps the cutting edge cool and ensures that the chips will be forced out through the chip groove. Cutting fluid is normally applied at a pressure of 3.4 to 4.1 MPa (500 to 600 psi) for drills 9.5 mm (⅜ in.) in diameter or smaller and at 2.1 to 2.8 MPa (300 to 400 psi) for larger drills. A pump of 3.7 kW (5 hp) capacity is usually required for average applications. A paraffin-base oil with a viscosity of 100 to 125 SUS at 40 °C (100 °F) has been used satisfactorily in gun drilling aluminum with a 9.5 mm (⅜ in.) diam drill at a speed of 90 m/min (300 sfm) for high-speed tool steel or 180 m/min (600 sfm) for carbide and at feeds of 0.025 to 0.08 mm/rev (0.001 to 0.003 in./rev).

Although deep-drilling problems have been alleviated to some extent by the development of gun drills and special gun drilling equipment, this equipment is expensive and

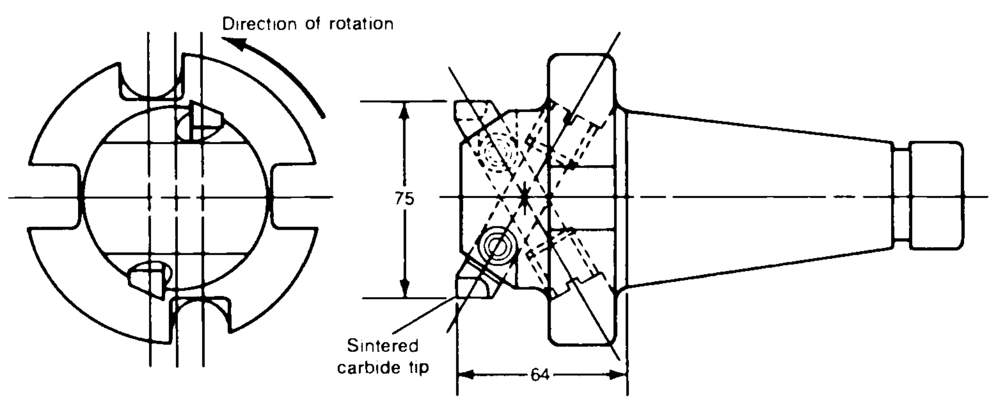


Fig. 24 Special two-cut tooth end mill with sintered carbide cutters. Dimensions given in millimeters. Source: Ref 4

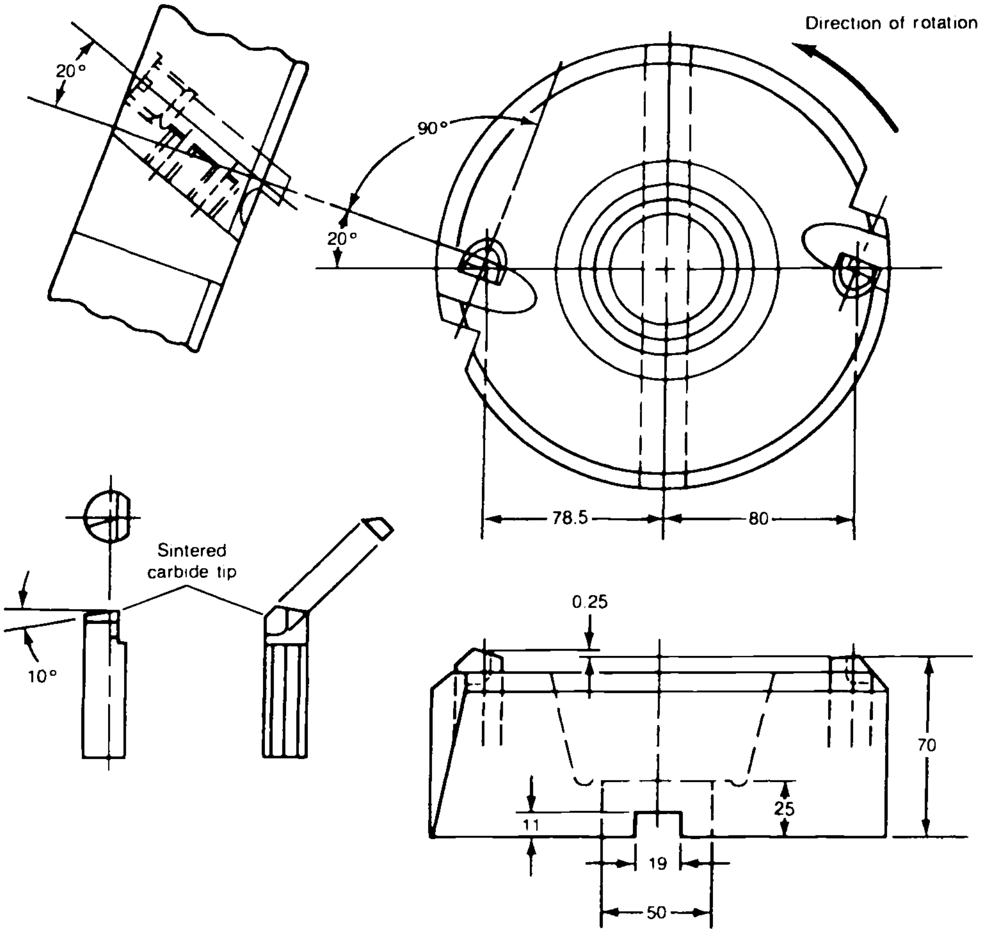


Fig. 25 Tool geometry for special two-tooth sintered carbide-tip milling cutter used to cut aluminum alloys. Dimensions given in millimeters. Source: Ref 4

cannot always be justified. The following example describes a deep-hole drilling application that utilized available equipment (in this case, a boring mill).

Example 4: Deep-Hole Drilling in a Boring Mill. The 914 mm (36 in.) diam, 914 mm (36 in.) long explosive forming die illustrated in Fig. 19 was originally made up from nine 102 mm (4 in.) thick aluminum alloy plates. Because no plate stock obtainable was wide enough to make a 914 mm (36 in.) diam one-piece plate, each round plate was made by doweling together two 102 mm (4 in.) thick half-sections. As each 102 mm (4 in.) plate section was completed by machining the die cavity hole through its center and drilling 12 tie-bolt holes, it was mated to the next 102 mm (4 in.) thick plate until the die was completed. Each individual plate was placed on the preceding plate so that the half-section cut lines were 90° from those in the preceding plate. Then each section was doweled with two pins to the next, completing the die.

By the improved method of producing these dies, four 914 mm (36 in.) diam, 229 mm (9 in.) thick forgings were obtained. These forgings were machined (as were the 102 mm, or 4 in., thick plates) individually and doweled to one another to maintain alignment. The assembly was then set up in a boring mill as shown in Fig. 19. Each of the 41 mm (1⅝ in.) diam tie-bolt holes was drilled in one pass using a standard gun drill. Drilling time was reduced from 7 h by the original method to ½ h by the improved method.

The boring mill was modified by adding a hydraulic pump with a capacity of 154 L/min (40.6 gal./min) to supply cutting fluid to the drill point at 2.1 MPa (300 psi). A baffle plate and filter system were installed to handle the flow of oil.

Reaming

The machines, basic designs of reamers, and techniques discussed in the article "Reaming" in this Volume are generally applicable to aluminum alloys.

Reamer Design. Because it is less likely to cause chatter, a spiral-flute reamer (solid, expandable, or adjustable) is generally preferable to a straight-flute reamer for finishing holes in aluminum alloys. In most applications, it is advantageous to use a reamer with a negative spiral (this is, one spiraled in the direction opposite to rotation) to prevent the reamer from feeding itself into the hole. Flutes should be large enough to pass the chips readily, and there should be enough flutes to provide adequate support to the tool. The margins of straight-flute reamers should be as narrow as possible to reduce friction between tool and work. Straight-flute reamers are designed with an even number of blades arranged opposite each other in pairs, but with flute spacing varied slightly to prevent chatter and marking.

Spiral flutes (right-hand cut, left-hand spiral) are frequently more effective than angular spacing in reducing chatter. However, a spiral-flute reamer must have sufficient spiral so that two or more flutes overlap in the length of the reamed hole. The spiral angle must be held to a minimum because the steeper the angle, the more end pressure is required to feed the reamer through the hole. Additional reamer design data are given in Table 20.

When the hole has close tolerances and rigid surface finish requirements, the reaming procedure must sometimes be altered. If the design of the workpiece permits, spiral-flute reamers with 7° hook will produce fine finishes, especially on angular surfaces. When reaming diameters of 19.05 mm (0.750 in.) or larger, a carbide-tip expandable reamer will produce good finish and provide extended tool life.

Speed and Feed. The nominal speed for reaming the nonheat-treated cast alloys with high-speed tool steel (M1, M2, or M7) reamers is 150 m/min (500 sfm). For all other cast and wrought alloys (excluding high-silicon alloys), speed should be about 120 m/min (400 sfm).

When reaming the nonheat-treated cast alloys with carbide tools, nominal speed is 300 m/min (1000 sfm). For other cast and wrought alloys (excluding high-silicon alloys), nominal reaming speed is 260 m/min (850 sfm).

Feed rate in reaming aluminum alloys is generally the same for all alloys and tool materials; hole size, however, does affect optimum feed, as indicated below:

Feed, mm/rev	Feed, in./rev	Hole diameter, mm	Hole diameter, in.
0.13	0.005	3.2	⅛
0.18	0.007	6.4	¼
0.30	0.012	13	½
0.38	0.015	25	1
0.51	0.020	38	1½
0.76	0.030	50	2

Table 21 lists nominal speeds and feeds for the rough and finish reaming of aluminum alloys.

Cutting Fluid. At high reaming speed, a cutting fluid is required for reducing the temperature in the workpiece, minimizing distortion, and preventing undersize reaming. For reaming with high-speed tool steel reamers, mixtures of lard oil and paraffin oil

Table 25 Nominal speeds and feeds for the peripheral end milling of aluminum alloys with high-speed tool steel and carbide tools

Material	Hardness, HB (500 kg)	Condition	Radial depth of cut(a) mm (in.)	High-speed steel tool: Speed, m/min (sfm)	Feed, mm/tooth (in./tooth) with a cutter diameter of: 10 mm (3/8 in.)	12 mm (1/2 in.)	18 mm (3/4 in.)	25–50 mm (1–2 in.)	Tool material grade, ISO (AISI)	Carbide tool: Speed, m/min (sfm)	Feed, mm/tooth (in./tooth) with a cutter diameter of: 10 mm (3/8 in.)	12 mm (1/2 in.)	18 mm (3/4 in.)	25–50 mm (1–2 in.)	Tool material grade, ISO (C)
Wrought															
EC 3005 6066	30–80	Cold drawn	0.5	245	0.075	0.102	0.13	0.18	S4, S5, S2	395	0.075	0.102	0.13	0.18	K20, M20
1060 4032 6070			(0.020)	(800)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(1300)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
1100 5005 6101			1.5	185	0.102	0.15	0.20	0.25		305	0.102	0.15	0.20	0.25	
1145 5050 6151			(0.060)	(600)	(0.004)	(0.006)	(0.008)	(0.010)		(1000)	(0.004)	(0.006)	(0.008)	(0.010)	
1175 5052 6253			diam/4	150	0.075	0.102	0.15	0.20		275	0.075	0.13	0.15	0.20	
1235 5056 6262			(diam/4)	(500)	(0.003)	(0.004)	(0.006)	(0.008)		(900)	(0.003)	(0.005)	(0.006)	(0.008)	
2011 5083 6463			diam/2	120	0.050	0.075	0.13	0.15		245	0.050	0.102	0.13	0.15	
2014 5086 6951			(diam/2)	(400)	(0.002)	(0.003)	(0.005)	(0.006)		(800)	(0.002)	(0.004)	(0.005)	(0.006)	
2017 5154 7001	75–150	Solution treated	0.5	245	0.075	0.102	0.13	0.18	S4, S5, S2	395	0.075	0.102	0.13	0.18	K20, M20
2018 5252 7004		and aged	(0.020)	(800)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(1300)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
2021 5254 7005			1.5	185	0.102	0.15	0.20	0.25		305	0.102	0.15	0.20	0.25	
2024 5454 7039			(0.060)	(600)	(0.004)	(0.006)	(0.008)	(0.010)		(1000)	(0.004)	(0.006)	(0.008)	(0.010)	
2025 5456 7049			diam/4	150	0.075	0.102	0.15	0.20		275	0.075	0.13	0.15	0.20	
2117 5457 7050			(diam/4)	(500)	(0.003)	(0.004)	(0.006)	(0.008)		(900)	(0.003)	(0.005)	(0.006)	(0.008)	
2218 5652 7075			diam/2	120	0.050	0.075	0.13	0.15		245	0.050	0.102	0.13	0.15	
2219 5657 7079			(diam/2)	(400)	(0.002)	(0.003)	(0.005)	(0.006)		(800)	(0.002)	(0.004)	(0.005)	(0.006)	
2618 6053 7175															
3003 6061 7178															
3004 6063															
Cast															
Sand and permanent mold															
A140 F332.0 520.0	40–100	As-cast	0.5	305	0.075	0.102	0.13	0.18	S4, S5, S2	395	0.075	0.102	0.15	0.18	K20, M20
201.0 333.0 535.0			(0.020)	(1000)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(1300)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
208.0 354.0 705.0			1.5	245	0.102	0.15	0.20	0.25		305	0.102	0.15	0.20	0.25	
213.0 355.0 707.0			(0.060)	(800)	(0.004)	(0.006)	(0.008)	(0.010)		(1000)	(0.004)	(0.006)	(0.008)	(0.010)	
222.0 C355.0 A712.0			diam/4	185	0.075	0.102	0.15	0.20		275	0.075	0.13	0.15	0.20	
224.0 356.0 D712.0			(diam/4)	(600)	(0.003)	(0.004)	(0.006)	(0.008)		(900)	(0.003)	(0.005)	(0.006)	(0.008)	
242.0 A356.0 713.0			diam/2	150	0.050	0.075	0.13	0.15		245	0.050	0.102	0.13	0.15	
295.0 357.0 771.0			(diam/2)	(500)	(0.002)	(0.003)	(0.005)	(0.006)		(800)	(0.002)	(0.004)	(0.005)	(0.006)	
B295.0 359.0 850.0	70–125	Solution treated	0.5	245	0.075	0.102	0.13	0.18	S4, S5, S2	395	0.075	0.102	0.13	0.18	K20, M20
308.0 B443.0 A850.0		and aged	(0.020)	(800)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(1300)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
319.0 514.0 B850.0			1.5	185	0.102	0.15	0.20	0.25		305	0.102	0.15	0.20	0.25	
328.0 A514.0			(0.060)	(600)	(0.004)	(0.006)	(0.008)	(0.010)		(1000)	(0.004)	(0.006)	(0.008)	(0.010)	
A332.0 B514.0			diam/4	150	0.075	0.102	0.15	0.20		275	0.075	0.13	0.15	0.20	
Hiduminium RR-350			(diam/4)	(500)	(0.003)	(0.004)	(0.006)	(0.008)		(900)	(0.003)	(0.005)	(0.006)	(0.008)	
			diam/2	120	0.050	0.075	0.13	0.15		245	0.050	0.102	0.13	0.15	
			(diam/2)	(400)	(0.002)	(0.003)	(0.005)	(0.006)		(800)	(0.002)	(0.004)	(0.005)	(0.006)	
Die castings															
360.0	40–100	As-cast	0.5	305	0.075	0.102	0.13	0.18	S4, S5, S2	395	0.075	0.102	0.13	0.18	K20, M20
A360.0			(0.020)	(1000)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(1300)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
380.0			1.5	245	0.102	0.15	0.20	0.25		305	0.102	0.15	0.20	0.25	
A380.0			(0.060)	(800)	(0.004)	(0.006)	(0.008)	(0.010)		(1000)	(0.004)	(0.006)	(0.008)	(0.010)	
C443.0			diam/4	185	0.075	0.102	0.15	0.20		275	0.075	0.13	0.15	0.20	
518.0			(diam/4)	(600)	(0.003)	(0.004)	(0.006)	(0.008)		(900)	(0.003)	(0.005)	(0.006)	(0.008)	
			diam/2	150	0.050	0.075	0.13	0.15		245	0.050	0.102	0.13	0.15	
			(diam/2)	(500)	(0.002)	(0.003)	(0.005)	(0.006)		(800)	(0.002)	(0.004)	(0.005)	(0.006)	
	70–125	Solution treated	0.5	275	0.075	0.102	0.13	0.18	S4, S5, S2	395	0.075	0.102	0.13	0.18	K20, M20
		and aged	(0.020)	(900)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(1300)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
			1.5	215	0.102	0.15	0.20	0.25		305	0.102	0.15	0.20	0.25	
			(0.060)	(700)	(0.004)	(0.006)	(0.008)	(0.010)		(1000)	(0.004)	(0.006)	(0.008)	(0.010)	
			diam/4	150	0.075	0.102	0.15	0.20		275	0.075	0.13	0.15	0.20	
			(diam/4)	(500)	(0.003)	(0.004)	(0.006)	(0.008)		(900)	(0.003)	(0.005)	(0.006)	(0.008)	
			diam/2	120	0.050	0.075	0.13	0.15		245	0.050	0.102	0.13	0.15	
			(diam/2)	(400)	(0.002)	(0.003)	(0.005)	(0.006)		(800)	(0.002)	(0.004)	(0.005)	(0.006)	
383.0	40–100	As-cast	0.5	275	0.075	0.102	0.13	0.18	S4, S5, S2	395	0.075	0.102	0.13	0.18	K20, M20
A384.0			(0.020)	(900)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(1300)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
413.0			1.5	215	0.102	0.15	0.20	0.25		305	0.102	0.15	0.20	0.25	
A413.0			(0.060)	(700)	(0.004)	(0.006)	(0.008)	(0.010)		(1000)	(0.004)	(0.006)	(0.008)	(0.010)	
			diam/4	150	0.075	0.102	0.15	0.20		275	0.075	0.13	0.15	0.20	
			(diam/4)	(500)	(0.003)	(0.004)	(0.006)	(0.008)		(900)	(0.003)	(0.005)	(0.006)	(0.008)	
			diam/2	120	0.050	0.075	0.13	0.15		245	0.050	0.102	0.13	0.15	
			(diam/2)	(400)	(0.002)	(0.003)	(0.005)	(0.006)		(800)	(0.002)	(0.004)	(0.005)	(0.006)	
	70–125	Solution treated	0.5	245	0.075	0.102	0.13	0.18	S4, S5, S2	305	0.075	0.102	0.13	0.18	K20, M20
		and aged	(0.020)	(800)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(1000)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
			1.5	185	0.102	0.15	0.20	0.25		275	0.102	0.15	0.20	0.25	
			(0.060)	(600)	(0.004)	(0.006)	(0.008)	(0.010)		(900)	(0.004)	(0.006)	(0.008)	(0.010)	
			diam/4	120	0.075	0.102	0.15	0.20		245	0.075	0.13	0.15	0.20	
			(diam/4)	(400)	(0.003)	(0.004)	(0.006)	(0.008)		(800)	(0.003)	(0.005)	(0.006)	(0.008)	
			diam/2	90	0.050	0.075	0.13	0.15		215	0.050	0.102	0.13	0.20	
			(diam/2)	(300)	(0.002)	(0.003)	(0.005)	(0.006)		(700)	(0.002)	(0.004)	(0.005)	(0.008)	

Table 25 (continued)

				High-speed steel tool						Carbide tool					
					Feed, mm/tooth (in./tooth) with a cutter diameter of:						Feed, mm/tooth (in./tooth) with a cutter diameter of:				
Material	Hardness, HB (500 kg)	Condition	Radial depth of cut(a) mm (in.)	Speed, m/min (sfm)	10 mm (3/8 in.)	12 mm (1/2 in.)	18 mm (3/4 in.)	25–50 mm (1–2 in.)	Tool material grade, ISO (AISI)	Speed, m/min (sfm)	10 mm (3/8 in.)	12 mm (1/2 in.)	18 mm (3/4 in.)	25–50 mm (1–2 in.)	Tool material grade, ISO (C)
390.0	40–100	As-cast	0.5	90	0.075	0.102	0.13	0.18	S4, S5, S2	185	0.075	0.102	0.13	0.18	K20, M20
392.0			(0.020)	(300)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(600)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
			1.5	60	0.102	0.15	0.20	0.25		150	0.102	0.15	0.20	0.25	
			(0.060)	(200)	(0.004)	(0.006)	(0.008)	(0.010)		(500)	(0.004)	(0.006)	(0.008)	(0.010)	
			diam/4	53	0.075	0.102	0.13	0.18		120	0.075	0.13	0.15	0.20	
			(diam/4)	(175)	(0.003)	(0.004)	(0.005)	(0.007)		(400)	(0.003)	(0.005)	(0.006)	(0.008)	
			diam/2	46	0.050	0.075	0.102	0.13		90	0.050	0.102	0.13	0.15	
			(diam/2)	(150)	(0.002)	(0.003)	(0.004)	(0.005)		(300)	(0.002)	(0.004)	(0.005)	(0.006)	
	70–125	Solution treated	0.5	76	0.075	0.102	0.13	0.18	S4, S5, S2	150	0.075	0.102	0.13	0.18	K20, M20
		and aged	(0.020)	(250)	(0.003)	(0.004)	(0.005)	(0.007)	(M2, M3, M7)	(500)	(0.003)	(0.004)	(0.005)	(0.007)	(C-2)
			1.5	60	0.102	0.15	0.20	0.25		120	0.102	0.15	0.20	0.25	
			(0.060)	(200)	(0.004)	(0.006)	(0.008)	(0.010)		(400)	(0.004)	(0.006)	(0.008)	(0.010)	
			diam/4	46	0.075	0.102	0.13	0.18		90	0.075	0.13	0.15	0.20	
			(diam/4)	(150)	(0.003)	(0.004)	(0.005)	(0.007)		(300)	(0.003)	(0.005)	(0.006)	(0.008)	
			diam/2	38	0.050	0.075	0.102	0.13		60	0.050	0.102	0.13	0.15	
			(diam/2)	(125)	(0.002)	(0.003)	(0.004)	(0.005)		(200)	(0.002)	(0.004)	(0.005)	(0.006)	

(a) For standard-length end mills, maximum axial depth can be up to 1.5 times the cutter diameter. Source: Ref 5

or kerosene, or of petroleum and turpentine, are especially recommended. Sulfurized and chlorinated oils are often used, but they are likely to stain the work. With carbide reamers, emulsions of soluble oil and water are the most widely used cutting fluids.

Reaming and Burnishing. When difficulty is encountered in obtaining the desired tolerance and surface finish, it may be necessary to employ two-stage finishing—for example, to ream the hole 0.013 mm (0.0005 in.) under the specified dimension and then to burnish it to size and finish requirements (see the article "Roller Burnishing" in this Volume.)

Reaming in Automatic Machines. A finishing reamer for use in automatic bar and chucking machines is shown in Fig. 20. This reamer finishes three diameters and faces the part to length. A radial rake angle of approximately 7° helps improve surface finish, particularly in reaming angular surfaces. The varied angular spacing of the flutes is intended to minimize slip, deflection, and chatter.

Burnishing

In the roller burnishing of aluminum, tools of hardened and polished steel are normally employed to finish a surface by compressing the surface while either the tool or the work is rotating. Holes in aluminum are also sometimes burnished in production by pressing a bearing-grade steel ball through the bore to improve the finish.

Tapping

Taps for producing threads in aluminum are usually made of one of the general-purpose grades of high-speed tool steel such as M1, M7, or M10. Taps should have polished flutes and ample backoff behind the land to prevent the pickup of work metal when the tap is withdrawn. Pitch diameters should be one thread class higher than those normally used for steel to compensate for elastic deformation during the tapping process.

Taps with a diameter less than 9.5 mm (3/8 in.) should have no more than two flutes. Larger taps should have the maximum number of flutes that will give the relationships of land width to circumference shown below:

Tap circumference, mm	Tap circumference, in.	Optimum land, %
13–25	1/2–1	33
25–44	1–1¾	25
>44	>1¾	20

The straight-flute tap is satisfactory for use with many aluminum alloys. The spiral-flute tap can be used for any of the alloys; this is better than the straight-flute type, especially for tapping soft material. A spiral-flute tap for cutting right-hand threads should have a right-hand spiral of about the same angle as that found on an ordinary twist drill; it should have a generous taper, and this taper should be backed off. The top rake on the front face should be approximately 25 to 45°. The rake on the back face enables the tap to cut rather than bind when it is reversed, and it provides a clean thread. The spiral-flute tap will bridge a keyway or slot in a hole and aid in the movement of chips from the hole.

Taps for through holes and the harder aluminum alloys should be provided with a hook angle of 10 to 20°, a spiral point, 3 to 4 threads chamfer, and a pitch diameter of GH2 (basic plus 0.013 to 0.025 mm, or 0.0005 to 0.001 in.) or GH3 (basic plus 0.025 to 0.038 mm, or 0.001 to 0.0015 in.). Taps for blind holes and soft alloys should have 10 to 15° hook and 40° right-hand spiral flutes.

For tapping blind holes less than 3.2 mm (1/8 in.) in diameter, spiral flutes should be avoided. For best size control, it is preferable not to bring the chip up the flute but to push it ahead of the tap with a spiral point. The efficiency of tapping may be greatly improved by a change in tooling or method—for example, a change from a solid tap to a collapsible tap.

The tapping speed recommended for aluminum is considerably higher than the speeds used for steel. Alloy composition and condition, thread pitch, and method of tapping are major factors affecting tapping speed.

Assuming that the thread pitch is fine (18 to 24) and that the operation can be closely controlled (as in leadscrew tapping), the nominal speeds given in Table 22 are generally suitable.

For coarser pitches, such as 8 to 12, any of the speeds given in Table 22 will be decreased because of the difficulty in controlling the machine. For short thread lengths, speeds for tapping a coarse thread should be about one-half the speeds given in Table 22. For long threads of the same pitch, the speed can be faster than for short threads because of the longer time between starting and stopping. The use of lead control devices permits higher tapping speeds than when no control is used. For example, a speed of 35 m/min (115 sfm) was used to tap 10-24 UNC threads in 2024-T4. The holes were blind and approximately 13 mm (1/2 in.) deep. However, leadscrew control was used. Under the same conditions but without leadscrew control, speed would have been 18 m/min (60 sfm) maximum. Table 23 is a more detailed listing of tapping speeds and tooling used.

Cutting Fluid. Lard oil diluted with kerosene or other low-viscosity mineral oil is usually preferred for tapping aluminum. Low-viscosity commercial cutting oils are also used successfully. Soluble-oil emul-

Table 26 Nominal speeds and feeds for the face milling of aluminum alloys with high-speed tool steel and carbide tools

Material	Hardness, HB (500 kg)	Condition	Depth of cut(a), mm (in.)	High-speed steel tool: Speed, m/min (sfm)	High-speed steel tool: Feed, mm/tooth (in./tooth)	High-speed steel tool: Tool material grade, ISO (AISI)	Carbide tool, uncoated: Speed: Brazed, m/min (sfm)	Carbide tool, uncoated: Speed: Indexable, m/min (sfm)	Carbide tool, uncoated: Feed, mm/tooth (in./tooth)	Carbide tool, uncoated: Tool material grade, ISO (C)
Wrought										
EC 3005 6066	30–80	Cold drawn	1	365	0.25	S4, S2	610	Max	0.25	K10, M20
1060 4032 6070			(0.040)	(1200)	(0.010)	(M2, M7)	(2000)	(Max)	(0.010)	(C-2)
1100 5005 6101			4	245	0.40	S4, S2	550	Max	0.50	K10, M20
1145 5050 6151			(0.150)	(800)	(0.015)	(M2, M7)	(1800)	(Max)	(0.020)	(C-2)
1175 5052 6253			8	200	0.50	S4, S2	365	Max	0.65	K20, M30
1235 5056 6262			(0.300)	(650)	(0.020)	(M2, M7)	(1200)	(Max)	(0.025)	(C-2)
2011 5083 6463	75–150	Solution treated	1	365	0.25	S4, S2	610	Max	0.25	K10, M20
2014 5086 6951		and aged	(0.040)	(1200)	(0.010)	(M2, M7)	(2000)	(Max)	(0.010)	(C-2)
2017 5154 7001			4	245	0.40	S4, S2	550	Max	0.50	K10, M20
2018 5252 7004			(0.150)	(800)	(0.015)	(M2, M7)	(1800)	(Max)	(0.020)	(C-2)
2021 5254 7005			8	200	0.50	S4, S2	365	Max	0.65	K20, M30
2024 5454 7039			(0.300)	(650)	(0.020)	(M2, M7)	(1200)	(Max)	(0.025)	(C-2)
2025 5456 7049										
2117 5457 7050										
2218 5652 7075										
2219 5657 7079										
2618 6053 7175										
3003 6061 7178										
3004 6063										
Cast										
Sand and permanent mold										
A140 F332.0 520.0	40–100	As-cast	1	365	0.20	S4, S2	610	760	0.25	K10, M20
201.0 333.0 535.0			(0.040)	(1200)	(0.008)	(M2, M7)	(2000)	(2500)	(0.010)	(C-2)
208.0 354.0 705.0			4	245	0.30	S4, S2	550	610	0.40	K20, M30
213.0 355.0 707.0			(0.150)	(800)	(0.012)	(M2, M7)	(1800)	(2000)	(0.015)	(C-2)
222.0 C355.0 A712.0			8	200	0.40	S4, S2	365	460	0.50	K30, M40
224.0 356.0 D712.0			(0.300)	(650)	(0.016)	(M2, M7)	(1200)	(1500)	(0.020)	(C-2)
242.0 A356.0 713.0	70–125	Solution treated	1	305	0.20	S4, S2	550	760	0.25	K10, M20
295.0 357.0 771.0		and aged	(0.040)	(1000)	(0.008)	(M2, M7)	(1800)	(2500)	(0.010)	(C-2)
B295.0 359.0 850.0			4	215	0.30	S4, S2	425	550	0.40	K20, M30
308.0 B443.0 A850.0			(0.150)	(700)	(0.012)	(M2, M7)	(1400)	(1800)	(0.015)	(C-2)
319.0 514.0 B850.0			8	170	0.40	S4, S2	305	395	0.50	K30, M40
328.0 A514.0			(0.300)	(550)	(0.016)	(M2, M7)	(1000)	(1300)	(0.020)	(C-2)
A332.0 B514.0										
Hiduminium RR-350										
Die castings										
360.0 A380.0	40–100	As-cast	1	365	0.20	S4, S2	425	550	0.13	K10, M20
A360.0 C443.0			(0.040)	(1200)	(0.008)	(M2, M7)	(1400)	(1800)	(0.005)	(C-2)
380.0 518.0			4	245	0.30	S4, S2	350	460	0.25	K10, M20
			(0.150)	(800)	(0.012)	(M2, M7)	(1150)	(1500)	(0.010)	(C-2)
	70–125	Solution treated	1	305	0.20	S4, S2	380	490	0.13	K10, M20
		and aged	(0.040)	(1000)	(0.008)	(M2, M7)	(1250)	(1600)	(0.005)	(C-2)
			4	215	0.30	S4, S2	305	395	0.25	K10, M20
			(0.150)	(700)	(0.012)	(M2, M7)	(1000)	(1300)	(0.010)	(C-2)
383.0	40–100	As-cast	1	305	0.20	S4, S2	350	460	0.13	K10, M20
A384.0			(0.040)	(1000)	(0.008)	(M2, M7)	(1150)	(1500)	(0.005)	(C-2)
413.0			4	215	0.30	S4, S2	290	365	0.25	K10, M20
A413.0			(0.150)	(700)	(0.012)	(M2, M7)	(950)	(1200)	(0.010)	(C-2)
	70–125	Solution treated	1	215	0.15	S4, S2	350	425	0.13	K10, M20
		and aged	(0.040)	(700)	(0.006)	(M2, M7)	(1150)	(1400)	(0.005)	(C-2)
			4	185	0.25	S4, S2	275	335	0.25	K10, M20
			(0.150)	(600)	(0.010)	(M2, M7)	(900)	(1100)	(0.010)	(C-2)
390.0	40–100	As-cast	1	50	0.15	S4, S2	130	150	0.13	K10, M20
392.0			(0.040)	(165)	(0.006)	(M2, M7)	(425)	(500)	(0.005)	(C-2)
			4	43	0.23	S4, S2	115	135	0.25	K10, M20
			(0.150)	(140)	(0.009)	(M2, M7)	(370)	(450)	(0.010)	(C-2)
	70–125	Solution treated	1	47	0.15	S4, S2	115	145	0.13	K10, M20
		and aged	(0.040)	(155)	(0.006)	(M2, M7)	(375)	(475)	(0.005)	(C-2)
			4	40	0.23	S4, S2	105	130	0.25	K10, M20
			(0.150)	(130)	(0.009)	(M2, M7)	(350)	(425)	(0.010)	(C-2)

(a) Depth of cut is measured parallel to axis of cutter. Source: Ref 5

Table 27 Nominal speeds and feeds for the face milling of aluminum alloys with diamond tools

Material				Hardness, HB (500 kg)	Condition	Depth of cut(a)		Speed		Feed	
						mm	in.	m/min	sfm	mm/tooth	in./tooth
Wrought											
EC	2218	5252	6253	30–150	All	0.25–0.75	0.010–0.030	610	2000	0.13	0.005
1060	2219	5254	6262			0.75–1.25	0.030–0.050	425	1400	0.25	0.010
1100	2618	5454	6463			1.25–2.50	0.050–0.100	305	1000	0.40	0.015
1145	3003	5456	6951								
1175	3004	5457	7001								
1235	3005	5652	7004								
2011	4032	5657	7005								
2014	5005	6053	7039								
2017	5050	6061	7049								
2018	5052	6063	7050								
2021	5056	6066	7075								
2024	5083	6070	7079								
2025	5086	6101	7175								
2117	5154	6151	7178								
Cast											
Sand and permanent mold											
A140	319.0	357.0	A712.0	40–100	As-cast	0.25–0.75	0.010–0.030	1005	3300	0.13	0.005
201.0	328.0	359.0	D712.0			0.75–1.25	0.030–0.050	840	2750	0.25	0.010
208.0	A332.0	B443.0	713.0			1.25–2.50	0.050–0.100	550	1800	0.40	0.015
213.0	F332.0	514.0	771.0	70–125	Solution treated and aged	0.25–0.75	0.010–0.030	915	3000	0.13	0.005
222.0	333.0	A514.0	850.0			0.75–1.25	0.030–0.050	760	2500	0.25	0.010
224.0	354.0	B514.0	A850.0			1.25–2.50	0.050–0.100	550	1800	0.40	0.015
242.0	355.0	520.0	B850.0								
295.0	C355.0	535.0									
B295.0	356.0	705.0									
308.0	A356.0	707.0									
Hiduminium RR-350											
Die castings											
360.0	A380.0			40–100	As-cast	0.25–0.75	0.010–0.030	1100	3600	0.13	0.005
A360.0	C443.0					0.75–1.25	0.030–0.050	975	3200	0.25	0.010
380.0	518.0					1.25–2.50	0.050–0.100	840	2750	0.40	0.015
383.0				40–100	As-cast	0.25–0.75	0.010–0.030	1005	3300	0.13	0.005
A384.0						0.75–1.25	0.030–0.050	840	2750	0.25	0.010
413.0						1.25–2.50	0.050–0.100	550	1800	0.40	0.015
A413.0											
390.0				40–100	As-cast	0.25–0.75	0.010–0.030	855	2800	0.13	0.005
392.0						0.75–1.25	0.030–0.050	670	2200	0.25	0.010
						1.25–2.50	0.050–0.100	470	1550	0.40	0.015

(a) Depth of cut is measured perpendicular to the axis of the cutter. Source: Ref 5

sions are sometimes used, but they will not provide as good a finish as the other cutting fluids mentioned. However, the following example describes an application in which a soluble oil was more effective than a mineral oil, probably because of the lower viscosity of the soluble-oil emulsion, which allowed it to penetrate more readily to the cutting edges of the tap and also allowed it to flow in greater volume for flushing away chips.

Example 5: Mineral Oil Versus Soluble Oil for Tapping. An automatic tapping machine was used for cutting 6-32 UNC-2B threads in through holes (6.4 mm, or ¼ in., deep) in alloy 2024-T4. Taps were straight-flute, spiral-point, with 15° chamfer angle and 18° hook, and they were operated at 4150 rev/min (37 m/min, or 122 sfm). Cutting fluid was mineral oil.

Chip congestion, together with buildup of work metal on the cutting edges of the tap, resulted in rough threads and tap breakage. Variations in tapping speed were tried but did not alleviate these problems. However, by substituting soluble oil (in a 1:20 mixture with water) for mineral oil as the cutting fluid, thread finish became acceptable, and productivity and tap life were increased:

	Mineral oil	Soluble oil
Holes tapped/h	1480	1520
Total tap life, holes	6250	8900

Optimum results are obtained when the cutting fluid is applied with pressure, especially when tapping blind holes.

Form Tapping. If chips are a serious problem and if the hole wall is thick enough to support the pressure of the tool, a form (chipless) tap can be used (see the article "Tapping" in this Volume). Although a 75% thread is generally recommended for cut threads, tap drilling for 55 to 65% thread is practical for threads produced with a form tap. All aluminum alloys except the high-silicon (12%) die casting alloys can be tapped with form taps.

In some applications, form taps have been successfully operated at speeds twice as fast as those used for their cutter-type counterparts. Limitations are the same for form tapping as for cutting tapping, and the same cutting fluids are used. The following example compares performance of cutting taps and form taps.

Example 6: Tapping: Cutting Versus Forming. The performance of No. 4-40 cutting taps was compared with that of form (chipless) taps in tapping a through hole in a 6.4 mm (¼ in.) section of alloy 380 die castings. Both taps were used in a leadscrew machine at 3000 rev/min. The cutting fluid was a high-grade cutting oil with low sulfur content. Although the production rate for both taps was 400 pieces per hour, the production rate per setup with the form tap (1200 pieces) was double the rate obtained when the cutting tap was used.

In tapping these die castings, a form tap was superior to a cutting tap in all sizes from No. 2 through No. 8, provided the hole wall was thick enough to support the pressure of the form tap.

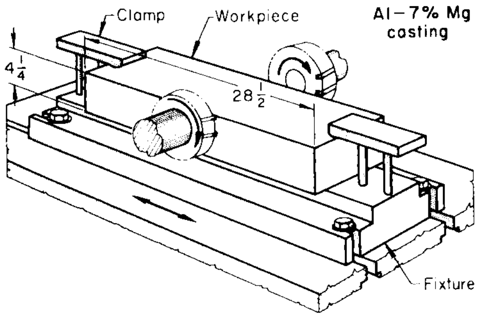
Table 28 Nominal speeds and feeds for the end milling and slotting of aluminum alloys with high-speed tool steel tools

Material	Hardness, HB (500 kg)	Condition	Axial depth of cut, mm (in.)	Speed, m/min (sfm)	Feed, mm/tooth (in./tooth), at a width of slot of: 10 mm (3/8 in.)	12 mm (1/2 in.)	18 mm (3/4 in.)	25–50 mm (1–2 in.)	High-speed tool steel material grade, ISO (AISI)
Wrought									
EC 2218 5254 6463	30–80	Cold drawn	0.75	150	0.075	0.13	0.15	0.25	S4, S5, S2
1060 2219 5454 6951			(0.030)	(500)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
1100 2618 5456 7001			3	135	0.102	0.15	0.20	0.30	
1145 3003 5457 7004			(0.125)	(450)	(0.004)	(0.006)	(0.008)	(0.012)	
1175 3004 5652 7005			diam/2	120	0.075	0.13	0.15	0.20	
1235 3005 5657 7039			(diam/2)	(400)	(0.003)	(0.005)	(0.006)	(0.008)	
2011 4032 6053 7049			diam/1	105	0.050	0.075	0.13	0.15	
2014 5005 6061 7050			(diam/1)	(350)	(0.002)	(0.003)	(0.005)	(0.006)	
2014 5050 6063 7075	75–150	Solution treated and aged	0.75	145	0.075	0.13	0.15	0.25	S4, S5, S2
2017 5052 6066 7079			(0.030)	(475)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
2018 5056 6070 7175			3	130	0.102	0.15	0.20	0.30	
2021 5083 6101 7178			(0.125)	(425)	(0.004)	(0.006)	(0.008)	(0.012)	
2024 5086 6151			diam/2	115	0.075	0.13	0.15	0.20	
2025 5154 6253			(diam/2)	(375)	(0.003)	(0.005)	(0.006)	(0.008)	
2117 5252 6262			diam/1	100	0.050	0.075	0.13	0.15	
			(diam/1)	(325)	(0.002)	(0.003)	(0.005)	(0.006)	
Cast									
Sand and permanent mold									
A140 319.0 357.0 A712.0	40–100	As-cast	0.75	120	0.075	0.13	0.15	0.25	S4, S5, S2
201.0 328.0 359.0 D712.0			(0.030)	(400)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
208.0 A332.0 B443.0 713.0			3	105	0.102	0.15	0.20	0.30	
213.0 F332.0 514.0 771.0			(0.125)	(350)	(0.004)	(0.006)	(0.008)	(0.012)	
222.0 333.0 A514.0 850.0			diam/2	90	0.075	0.13	0.15	0.20	
224.0 354.0 B514.0 A850.0			(diam/2)	(300)	(0.003)	(0.005)	(0.006)	(0.008)	
242.0 355.0 520.0 B850.0			diam/1	76	0.050	0.075	0.13	0.15	
295.0 C355.0 535.0			(diam/1)	(250)	(0.002)	(0.003)	(0.005)	(0.006)	
B295.0 356.0 705.0	70–125	Solution treated and aged	0.75	115	0.075	0.13	0.15	0.25	S4, S5, S2
308.0 A356.0 707.0			(0.030)	(375)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
Hiduminium RR-350			3	100	0.102	0.15	0.20	0.30	
			(0.125)	(325)	(0.004)	(0.006)	(0.008)	(0.012)	
			diam/2	84	0.075	0.13	0.15	0.20	
			(diam/2)	(275)	(0.003)	(0.005)	(0.006)	(0.008)	
			diam/1	69	0.050	0.075	0.13	0.15	
			(diam/1)	(225)	(0.002)	(0.003)	(0.005)	(0.006)	
Die castings									
360.0	40–100	As-cast	0.75	150	0.075	0.13	0.15	0.25	S4, S5, S2
A360.0			(0.030)	(500)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
380.0			3	135	0.102	0.15	0.20	0.30	
A380.0			(0.125)	(450)	(0.004)	(0.006)	(0.008)	(0.012)	
C443.0	70–125	Solution treated and aged	0.75	145	0.075	0.13	0.15	0.25	S4, S5, S2
518.0			(0.030)	(475)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
			3	130	0.102	0.15	0.20	0.30	
			(0.125)	(425)	(0.004)	(0.006)	(0.008)	(0.012)	
383.0	40–100	As-cast	0.75	135	0.075	0.13	0.15	0.25	S4, S5, S2
A384.0			(0.030)	(450)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
413.0			3	130	0.102	0.15	0.20	0.30	
A413.0			(0.125)	(425)	(0.004)	(0.006)	(0.008)	(0.012)	
	70–125	Solution treated and aged	0.75	130	0.075	0.13	0.15	0.25	S4, S5, S2
			(0.030)	(425)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
			3	120	0.102	0.15	0.20	0.30	
			(0.125)	(400)	(0.004)	(0.006)	(0.008)	(0.012)	
390.0	40–100	As-cast	0.75	53	0.075	0.13	0.15	0.25	S4, S5, S2
392.0			(0.030)	(175)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
			3	46	0.102	0.15	0.20	0.30	
			(0.125)	(150)	(0.004)	(0.006)	(0.008)	(0.012)	
	70–125	Solution treated and aged	0.75	46	0.075	0.13	0.15	0.25	S4, S5, S2
			(0.030)	(150)	(0.003)	(0.005)	(0.006)	(0.010)	(M2, M3, M7)
			3	38	0.102	0.15	0.20	0.30	
			(0.125)	(125)	(0.004)	(0.006)	(0.008)	(0.012)	

Source: Ref 5

Drilling and tapping of high-silicon alloy 390 (Ref 2) can be successfully accomplished with the same tool materials and geometries used for other aluminum casting alloys; however, cutting must be performed at lower speeds. Selection of the best tool material and geometry becomes more significant as the hole-depth-to-diameter ratio increases.

The use of natural or synthetic soluble-oil cutting fluids is strongly recommended. Coolant should be directed into the hole with a volume and pressure sufficient to reach the cutting lips of the tool and to flush the chips out of the hole. Inadequate coolant can result in chips packing in the flutes of a drill or the teeth of a tap, a condition that can cause rapid wear, poor hole finish,



Cutter details

Type	Face mill, inserted blade
Size, mm (in.)	152 (6) diameter
Number of teeth	12
Material	Carbide-tip blades

Operating conditions

Speed, at 2345 rev/min, m/min (sfm)	1120 (3680)
Feed, mm/tooth (in./tooth)	0.14 (0.0055)
Depth of cut, mm (in.)	3.2 (⅛) max
Cutting fluid	None
Setup time, h	1.5
Downtime for changing tools, min	15
Production rate, pieces/h	31
Cutter life, pieces/grind	~100
Tolerances, mm (in.)	
Flatness	Within 0.025 (0.001)
Width	±0.08 (±0.003)
Parallelism error	0.13 (0.005) max
Finish, µm (µin.)	1.60 (63)

Fig. 26 Simultaneous face milling of parallel surfaces on opposite sides of a casting. Dimensions in figure given in inches

and blowout fracturing of the exit surface upon emergence of a through hole drill. Even more important with respect to alloy 390 is that the chips are exceptionally abrasive and can wear on the lip and margin of a drill if not quickly flushed from the hole.

In a recent study, several drill and tap materials and geometries were evaluated, and cutting conditions were optimized for the most promising combinations. Chip packing became a problem at a depth of approximately three diameters (13 mm, or ½ in., holes), with slow and regular helix drills (24 to 33°) causing breakthrough blowouts and rapid drill wear. Solid high-speed tool steel drills with a high helix angle (38°) and a parabolic flute shape showed less wear under all cutting conditions than high-helix (40°) drills that had a standard flute design.

A speed of 20 m/min (70 sfm) and a feed rate of 381 mm/min (15 in./min) worked well with a high-helix, parabolic flute shaped drill (Fig. 21). A tool life of over 1000 holes was achieved when drilling 13 mm (½ in.) holes to a depth of five to six diameters. Heavy feed rate (in terms of inches per revolution) produced a better tool life than light feed rate.

The best method of drilling alloy 390 is to move the chips quickly by flushing with coolant and by moving the tool through the work in as few revolutions as possible. This creates an easier-to-remove, substantial chip.

Coolant-fed drills were also evaluated, but with mixed results. Carbide-tip, coolant-fed drills produced 1000 holes at a speed of 90 m/min (300 sfm) and a feed rate of 508 mm/min (20 in./min) (Fig. 21). Not only did this accomplish a much higher production rate, but hole size control, roundness, straightness, and finish were much improved over noncoolant-fed drills. Drilling with coolant-fed, high-speed tool steel, on the other hand, resulted in no productivity or hole quality improvement over the best solid drills.

Far better tool life results have been reported using diamond-tip gun drills in similar-size holes (10.79 mm, or 0.425 in.) at a hole depth of eight diameters when boring through solid material. When used in the mass production of automobile engine cylinders, these diamond-tip gun drills give a service life 60 times that of customary twist drills.

Tapping tests were conducted with various types of thread cutting and thread forming tools. No thread cutting taps threaded more than 360 holes before they would no longer gage to size. The best performance, more than 1000 holes, was achieved with a standard form tap having four lobes. It has also been shown that form taps produce the strongest threads. The conditions that were most successful for both the drilling and tapping experiments are given in Table 24.

Single-Point Threading

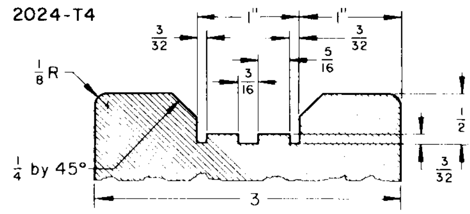
Single-point threading tools of conventional design are used to cut both internal and external threads on aluminum alloys. Speeds of 150 m/min (500 sfm) are common, although higher speeds have been used. Typical practice is represented by the following example.

Example 7: Single-Point Threading of Die Castings. A fine-pitch thread (pitch diameter: 42.113/41.946 mm, or 1.6580/1.6514 in.) was machined in an alloy 380 die casting (Fig. 22) with a single-point carbide tool. This alloy was susceptible to chipping and tearing if high cutting pressure was applied, particularly if the die casting was porous in the area being threaded.

Die Threading

Circular chasers for die threading aluminum alloys should have a hook angle of 20° and a face angle of 2°. When threading with tangential chasers, a combination of nominal back rake angle of 20° and side rake angle of 0° is generally best.

To avoid damage to the first few threads, chasers should have a lead chamfer of 25 to 35° for 1½ threads. This chamfer will ensure a smooth, even start.



Operating conditions(a)

Form milling (one pass), mm/min (in./min)	75 (3)
Standard milling (five passes), mm/min (in./min)	
1 Side mill slot	203 (8)
2 Side mill center slot	203 (8)
3 Straddle mill reliefs	127 (5)
4 Straddle mill two 6.4 mm (¼ in.) × 45° chamfers	203 (8)
5 Straddle mill two 3.2 mm (⅛ in.) radii	203 (8)

Time analysis

Time/piece, min	
Form milling	2.54
Standard milling	7.79
Time saved by using form milling	5.25

(a) For all operations, cutter speed was 23 m/min (75 sfm), and sulfurized oil was used as the cutting fluid.

Fig. 27 Contour that could be milled in a single pass using a form cutter or in five operations using standard milling cutters. Dimensions in figure given in inches

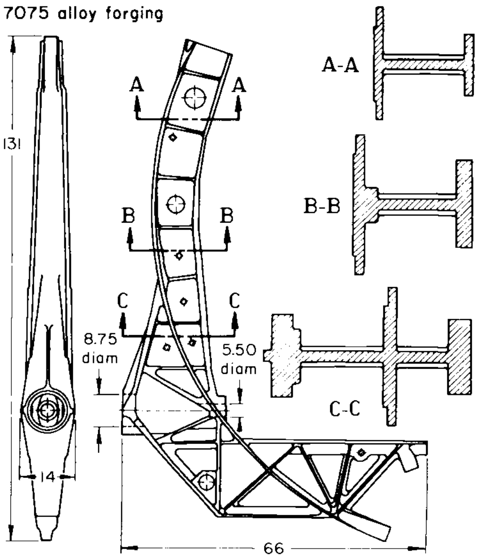
Speeds up to 40 m/min (130 sfm) for the nonheat-treated cast alloys and about 30 m/min (100 sfm) for the other alloys can be used, if the length and pitch of the thread and the equipment used permit control at these speeds. Speeds no more than half of the above are more often used because at the higher speeds control is more difficult, especially when threading short lengths or close to a shoulder.

Milling

The characteristics of chip formation sometimes cause difficulty in the milling of aluminum alloys. In some applications, chip ejection in the milling of deep slots or heavy cuts can be improved by changing from an alloy in the O or F temper to one in the T4 or T6 temper. In the T tempers, the chip is much less likely to clog the cutter. However, when aluminum is milled in the heat-treated condition, the production rate may be lower because a lighter feed rate may be necessary.

Power Requirements. Metal removal rates greater than 2600 mm³/min/kW (3.0 in.³/min/hp) can be readily attained in the production milling of aluminum alloys. Some rules applicable to milling aluminum are:

- If feed per tooth is doubled, the horsepower at the cutter must be increased in the ratio of 3:2, or 50%
- If the width of cut is doubled, the horsepower at the cutter must be doubled



Machine	Three-axis profile milling machine, numerically controlled, 4370 × 1320 × 457 mm (172 × 52 × 18 in.)
Cutter	
Material	Modified M2 high-speed tool steel
Number of cutters	27
Size of cutters, mm (in.)	19.1 (0.75) diameter × 95 (3¾) long to 76.2 (3.00) diameter × 203 (8) long
Rake angles	10° radial, 30° axial
Operating conditions	
Speed, m/min (sfm)	60–135 (200–450)
Feed, mm/tooth (in./tooth)	0.15–0.33 (0.006–0.013)
Depth of cut, axial dimension, mm (in.)	3.0–178 (0.12–7.00)
Width of cut, radial dimension, mm (in.)	9.40 (0.37) to cutter diameter
Machining time, each part, h	91(a)
Cutting fluid	Soluble oil:water (1:25)
Tolerance on critical dimensions, mm (in.)	±0.05 (±0.002)
Surface finish, μm (μin.)	1.50–3.05 (60–120)

(a) Roughing, 29 h; finishing, 62 h

Fig. 28 Part that was machined from a large blocker forging by NC profile milling. Dimensions in figure given in inches

- If the depth of cut is doubled, the horsepower at the cutter must be increased in the ratio of 1.9:1, or 90%
- If the depth of cut is halved and horsepower at the cutter remains the same, the feed may be increased by a factor of 2.5
- If the speed of the cutter is doubled and the feed per tooth is halved, the horsepower at the cutter must be increased by about 30%

Cutter Design. For efficiency in milling aluminum, the cutter should have a radial rake angle of 10 to 20°, an axial rake angle of 15 to 45°, and end or peripheral clearance of 10 to 12°. Form-relieved cutters should have a relief of 10° on the profile.

Best results can be obtained with a cutter that has fewer teeth and larger positive-rake angles than cutters for milling steel. Fewer teeth permit more chip space, which is especially important in milling aluminum because speeds are usually much greater than those used in milling steel. To prevent chatter, however, the cutter should have enough teeth so that at least two teeth are engaged at all times.

Because of the high speeds used in milling aluminum, careful consideration must be given to the effect of centrifugal force on the cutter. This is more important with cutters of large diameter. Cutter bodies in which teeth are wedge locked and from which some of the teeth have been removed to provide greater chip space should not be operated at high speed. All cutters for high-speed milling that have inserted teeth (brazed or mechanically secured) should be dynamically balanced. Careful attention should be given to cutter adapters, cutter holders, arbors, spacers, and other components that rotate with the cutter.

End mills for aluminum should have sharp rake angles with sufficient clearance to prevent heeling. Neither rake nor clearance angles should be excessive, or digging-in will result. The cutter should contact the workpiece in such a manner that any torsional, or lateral, deflection will decrease the depth of cut.

Flute form and surface finish in the flute are important. Consequently, many of these cutters are made by grinding from the solid after heat treatment. The lips of an end mill (of either the plain two-flute or ball-nose type) that is to be used for making plunge cuts directly into solid aluminum should have slightly greater clearances than is normal for these cutters in order to prevent heeling. Preferably, the faces of the cutting edges should have a surface finish of 0.038 μm (1.5 μin.) and should never be rougher than 0.088 μm (3.5 μin.).

In making plunge cuts directly into aluminum, the problem of chip ejection becomes acute. For this reason, it is best to feed the cutter into the material in such a way that, for each axial advance of the cutter (equal to about half the cutter diameter), the cutter is fed laterally about one diameter in an oscillating pattern until the desired depth of cut is reached, rather than making a purely axial cut.

For milling slots, or pockets, where the end of the cutter is in contact with the workpiece, it is usually best to have the same direction for the cut and the helix. However, when profiling with the periphery of the cutter, where the end of the cutter is not in contact with the workpiece, a combination of right-hand cut, left-hand helix gives the best results.

Router bits are a special type of end mill that is modified to ensure chip ejection at the speeds normally used for these cutters.

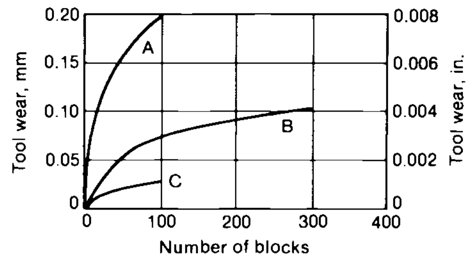


Fig. 29 Tool wear curves for the single-tooth milling of alloy 390 engine blocks (wet) at 0.30 mm/rev (0.012 in./rev). A, carbide, 150 m/min (492 sfm); B, diamond, 1500 m/min (4920 sfm); C, diamond, 150 m/min (492 sfm). Source: Ref 2

Some cutters for general routing purposes may have two flutes with a helix angle of 25°. The best overall performance in routing accurate slots or grooves is with a two-flute cutter having a 45° helix angle, with the direction of helix and of cut the same.

Cutters for routing stacked sheet stock should be designed with a single flute and a helix angle of 25 to 45°. An integral pilot running in an outboard bearing is used on some of these cutters to help control deflection.

All cutters should have a 15° hook angle on the cutting lip. The flute should be of 2.3 mm (0.09 in.) uniform depth, with a smooth gullet extending 3.2 mm (⅛ in.) inward from the lip. The lips should be radially relieved on the periphery to 0.08 to 0.18 mm (0.003 to 0.007 in.) per 1.6 mm (1⁄16 in.) width of land behind the cutting edge.

Special-Geometry End Mills for Aluminum (Ref 1, 4). An interesting example of fluted cutters designed specifically for machining aluminum is provided by the end mills shown in Fig. 23. The first departure from standard end-mill design is an increase to 45° in the helix angle from the standard 30°—the equivalent of high, positive side rake. The second departure is in the form of the tooth, the face of which provides a curved, positive-rake, polished primary surface followed by a secondary undercut that blends into the web (Fig. 23). The polished primary face surface and the discontinuity provide smooth chip flow and minimize the adherence of chips that invariably leads to clogging.

Two versions of these aluminum cutting end mills are available—one for heavy metal removal applications (such as pocketing and rough profiling) and one for finishing operations in which the radial depth of cut approximates 5% of tool diameter. The latter type incorporates an unrelieved circular land of 0.08 to 0.13 mm (0.003 to 0.005 in.) width on the periphery to minimize any tendency toward chatter, vibration, or squealing.

Somewhat similar to this concept is the use of a molded chip-breaker style of carbide insert. This type of insert increases the positive rake (nominally 5° rake, with 5°

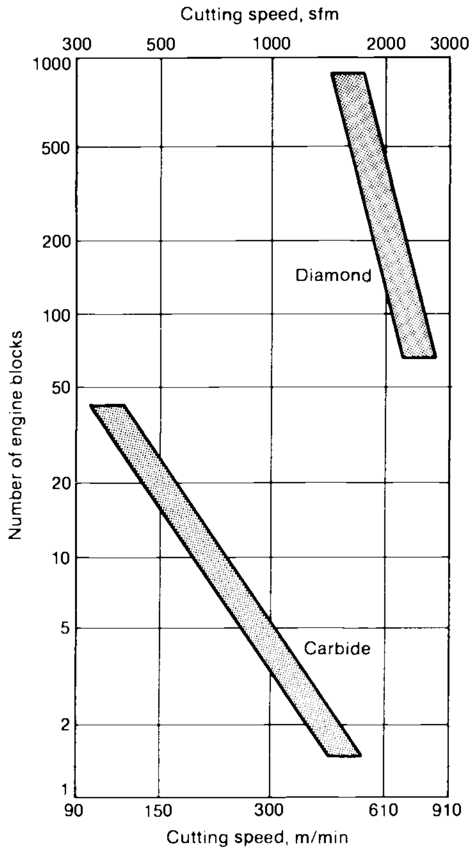


Fig. 30 Tool life of carbide and diamond tools when milling alloy 390 engine blocks (dry). Source: Ref 2

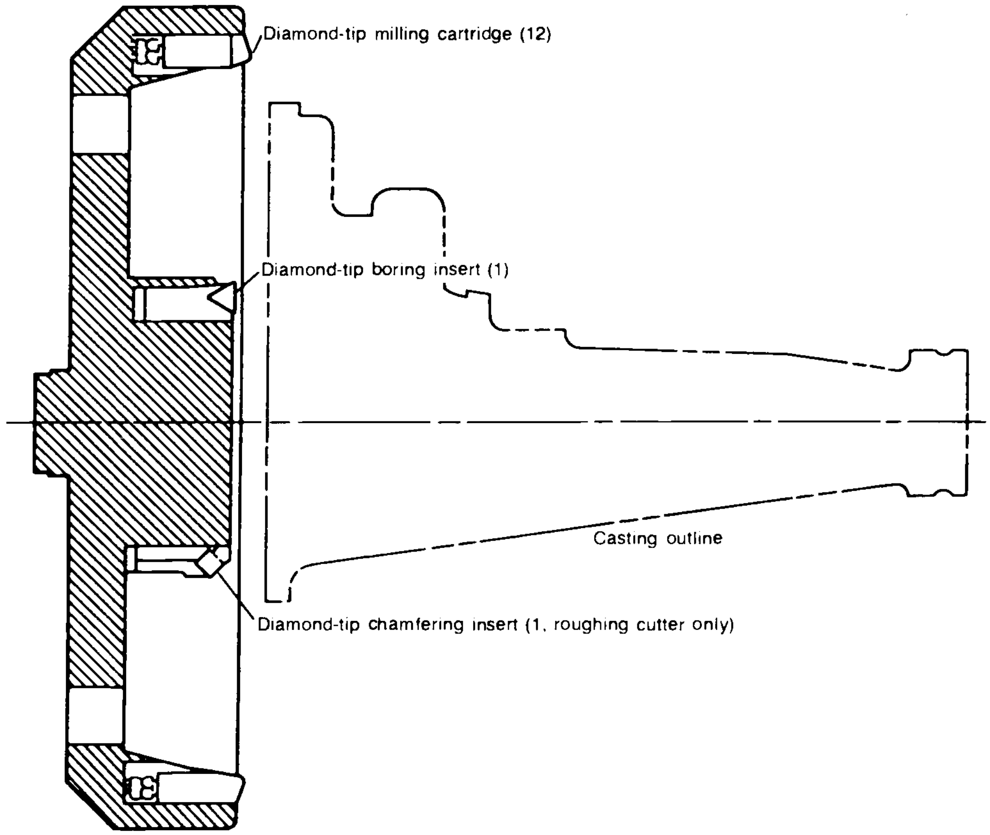


Fig. 31 Schematic of a combination boring-milling cutter with diamond-tip cutters machining a 380 aluminum alloy transmission extension housing at 3840 m/min (12 600 sfm)

clearance in the matching toolholder) while maintaining stronger edge geometry.

Although the most common geometry for positive-rake inserts provides a relief of 10 to 11°, which is decreased by the top rake angle at which it is held, it is also possible to specify carbide inserts with higher relief angles. One supplier of these inserts recommends a 20° relief insert for turning applications and 26° for milling—and without any chip control groove, especially for the gummier alloys. This allows the top surface of the insert to be lapped or polished to the high finish desirable for improved chip flow.

Special end mills and face mills designed with two carbide inserts for high milling speeds can be used to reduce milling time. Cutters for operations at high speeds should be statically and dynamically balanced.

The carbide inserts in the cutter shown in Fig. 24 can be adjusted for the depth of cut and the most effective rake angle. The radial and axial rake angles of the end milling cutter shown in Fig. 25 result from the position of the insert in the body of the cutter. This cutter removed a large amount of aluminum and incorporates adequate chip space.

The number of teeth in a cutter is largely determined by its diameter, the depth of the cut, and the type of cut (rough or finishing cut). Various formulas have been developed by cutter manufacturers to determine the optimum number of teeth for all types and sizes of milling cutters for milling aluminum. For good cutter performance, as a general rule, not more than two of the teeth should engage the cut at the same time. The teeth should have a strong cross section but ample chip space.

Speed. Nominal speeds for peripheral, face (high-speed, carbide, and diamond), and end milling are given in Tables 25 to 28. Factors affecting speed are the alloy being milled, tool material, and depth of cut (roughing or finishing). A major variable that is not reflected in Tables 25 to 28 is chip disposal. If the chips can be adequately ejected from the cutting zone, speeds for carbide cutters are limited mainly by the capabilities of the machine. However, when the speed is too high for a particular setup, the cutter teeth may not have sufficient time to remove stock at the proper rate because of erratic feeding; the cutter teeth may dig and ride, or cut and skip. Optimum speed depends on the machine, work metal, depth of cut, and power.

Under certain conditions, milling speeds as high as 4600 m/min (15 000 sfm) have been used. However, speeds this high are rarely feasible. Speeds of 600 to 1200 m/min (2000 to 4000 sfm) are frequently used, as illustrated in the following example.

Example 8: Face Milling Parallel Sides Simultaneously in a Duplex Machine. Face milling two opposite sides was the first machining operation performed on the casting shown in Fig. 26. Maintaining required dimensions in subsequent operations depended greatly on the accuracy obtained in milling the sides. Parallelism was easily maintained by milling the two sides simultaneously because the workpiece did not have to be moved and reclamped.

A 7.5 kW (10 hp) duplex machine was used for this operation. This type of machine afforded two advantages in addition to good dimensional control. First, after the initial cut, the spindles were advanced slightly to take a skim (finish) cut as the table returned to the start position. Second, milling both sides at one time increased productivity, compared to milling each side separately. Details of the milling cutters and operation are tabulated below Fig. 26.

Feed. Nominal feeds for peripheral, face, and end milling are given in Tables 25 to 28. For wheel-type cutters (peripheral and face), the alloy being milled has little effect on feed. Depth of cut has some effect, although usually this is not large. Some reduction in feed is usually made when high-speed tool steel cutters are replaced by

Table 29 Nominal speeds and feeds for circular sawing of aluminum alloys with high-speed tool steel blades

Material	Hardness, HB (500 kg)	Condition	Solid stock diameter or thickness, mm (in.)	Pitch, mm/tooth (in./tooth)	Cutting speed, m/min (sfm)	Feed, mm/tooth (in./tooth)	High-speed tool steel material grade, ISO (AISI)
Wrought							
EC 2218 5252 6253	30–80	Cold drawn	6–80	5–30	305	0.25	S4, S2
1060 2219 5254 6262			(¼–3)	(0.20–1.10)	(1000)	(0.010)	(M2, M7)
1100 2618 5454 6463			80–160	18–40	245	0.25	
1145 3003 5456 6951			(3–6)	(0.70–1.50)	(800)	(0.010)	
1175 3004 5457 7001			160–250	25–45	185	0.30	
1235 3005 5652 7004			(6–9)	(1.00–1.80)	(600)	(0.012)	
2011 4032 5657 7005			250–400	30–60	150	0.30	
2014 5005 6053 7039			(9–15)	(1.20–2.40)	(500)	(0.012)	
2017 5050 6061 7049	75–150	Solution treated and aged	6–80	5–30	305	0.25	S4, S2
2018 5052 6063 7050			(¼–3)	(0.20–1.10)	(1000)	(0.010)	(M2, M7)
2021 5056 6066 7075			80–160	18–40	245	0.25	
2024 5083 6070 7079			(3–6)	(0.70–1.50)	(800)	(0.010)	
2025 5086 6101 7175			160–250	25–45	185	0.30	
2117 5154 6151 7178			(6–9)	(1.00–1.80)	(600)	(0.012)	
			250–400	30–60	150	0.30	
			(9–15)	(1.20–2.40)	(500)	(0.012)	
Cast							
Sand and permanent mold							
A140 319.0 357.0 A712.0	40–100	As-cast	6–80	5–30	380	0.25	S4, S2
201.0 328.0 359.0 D712.0			(¼–3)	(0.20–1.10)	(1250)	(0.010)	(M2, M7)
208.0 A332.0 B443.0 713.0			80–160	18–40	305	0.25	
213.0 F332.0 514.0 771.0			(3–6)	(0.70–1.50)	(1000)	(0.010)	
222.0 333.0 A514.0 850.0			160–250	25–45	230	0.30	
224.0 354.0 B514.0 A850.0			(6–9)	(1.00–1.80)	(750)	(0.012)	
242.0 355.0 520.0 B850.0			250–400	30–60	185	0.30	
295.0 C355.0 535.0			(9–15)	(1.20–2.40)	(600)	(0.012)	
B295.0 356.0 705.0	70–125	Solution treated and aged	6–80	5–30	305	0.25	S4, S2
308.0 A356.0 707.0			(¼–3)	(0.20–1.10)	(1000)	(0.010)	(M2, M7)
Hiduminium RR-350			80–160	18–40	245	0.25	
			(3–6)	(0.70–1.50)	(800)	(0.010)	
			160–250	25–45	185	0.30	
			(6–9)	(1.00–1.80)	(600)	(0.012)	
			250–400	30–60	150	0.30	
			(9–15)	(1.20–2.40)	(500)	(0.012)	
Die castings							
360.0 A380.0	40–100	As-cast	6–80	5–30	380	0.25	S4, S2
A360.0 C443.0			(¼–3)	(0.20–1.10)	(1250)	(0.010)	(M2, M7)
380.0 518.0	70–125	Solution treated and aged	6–80	5–30	305	0.25	S4, S2
			(¼–3)	(0.20–1.10)	(1000)	(0.010)	(M2, M7)
383.0	40–100	As-cast	6–80	5–30	380	0.25	S4, S2
A384.0			(¼–3)	(0.20–1.10)	(1250)	(0.010)	(M2, M7)
413.0	70–125	Solution treated and aged	6–80	5–30	305	0.25	S4, S2
A413.0			(¼–3)	(0.20–1.10)	(1000)	(0.010)	(M2, M7)
390.0	40–100	As-cast	6–80	5–30	150	0.25	S4, S2
392.0			(¼–3)	(0.20–1.10)	(500)	(0.010)	(M2, M7)
	70–125	Solution treated and aged	6–80	5–30	150	0.25	S4, S2
			(¼–3)	(0.20–1.10)	(500)	(0.010)	(M2, M7)

Source: Ref 5

Table 30 Nominal speeds and feeds for the circular sawing of aluminum alloys with carbide-tip tools

Quantity of available data was limited. Performance data are presented instead of the general recommendations given in other machining operations in this Volume.

Hardness, HB (500 kg)	Condition	Solid stock diameter or thickness		Pitch		Feed		Cutting speed		Carbide tool material grade	
		mm	in.	mm/tooth	in./tooth	mm/tooth	in./tooth	m/min	sfm	ISO	C
<60	Cold drawn	6–25	0.25–1	15–25	0.60–1.00	0.25	0.010	4575	15 000	K30	C-2
		25–50	1–2	23–35	0.90–1.40	0.25	0.010	4575	15 000		
		50–100	2–4	25–50	1.00–2.00	0.13	0.005	3650	12 000		
		100–150	4–6	40–65	1.50–2.50	0.13	0.005	3650	12 000		
60–150	Solution treated and aged	6–25	0.25–1	15–25	0.60–1.00	0.18	0.007	3650	12 000	K30	C-2
		25–50	1–2	23–35	0.90–1.40	0.18	0.007	3650	12 000		
		50–100	2–4	25–50	1.00–2.00	0.075	0.003	3050	10 000		
		100–150	4–6	40–65	1.50–2.50	0.075	0.003	3050	10 000		
>150	Solution treated and aged	6–25	0.25–1	15–25	0.60–1.00	0.13	0.005	3050	10 000	K30	C-2
		25–50	1–2	23–35	0.90–1.40	0.13	0.005	3050	10 000		
		50–100	2–4	25–50	1.00–2.00	0.050	0.002	2750	9 000		
		100–150	4–6	40–65	1.50–2.50	0.050	0.002	2750	9 000		

Source: Ref 5

Table 31 Nominal speeds for the power band sawing of aluminum alloys with high-speed tool steel blades

Material				Hardness, HB (500 kg)	Condition	Material thickness		Tooth form(a)	Thread designation		Band speed	
						mm	in.		Pitch, mm	Teeth/in.	m/min	sfm
Wrought												
EC	2218	5252	6253	30–80	Cold drawn	<12	<½	P	1.8–1.4	14–18	305(b)	1000(b)
1060	2219	5254	6262			12–25	½–1	P	2.5–1.8	10–14	275(b)	900(b)
1100	2618	5454	6463			25–75	1–3	P	4–2.5	6–10	230(b)	750(b)
1145	3003	5456	6951			>75	>3	C	8.5	3	200(b)	650(b)
1175	3004	5457	7001	75–150	Solution treated and aged	<12	<½	P	1.8–1.4	14–18	275(b)	900(b)
1235	3005	5652	7004			12–25	½–1	P	2.5–1.8	10–14	245(b)	800(b)
2011	4032	5657	7005			25–75	1–3	P	4–2.5	6–10	200(b)	650(b)
2014	5005	6053	7039			>75	>3	C	8.5	3	170(b)	550(b)
2017	5050	6061	7049									
2018	5052	6063	7050									
2021	5056	6066	7075									
2024	5083	6070	7079									
2025	5086	6101	7175									
2117	5154	6151	7178									
Cast												
Sand and permanent mold												
A140	319.0	357.0	A712.0	40–100	As-cast	<12	<½	P	1.8–1.4	14–18	305(b)	1000(b)
201.0	328.0	359.0	D712.0			12–25	½–1	P	2.5–1.8	10–14	275(b)	900(b)
208.0	A332.0	B443.0	713.0			25–75	1–3	P	4–2.5	6–10	230(b)	750(b)
213.0	F332.0	514.0	771.0			>75	>3	C	8.5	3	200(b)	650(b)
222.0	333.0	A514.0	850.0	70–125	Solution treated and aged	<12	<½	P	1.8–1.4	14–18	275(b)	900(b)
224.0	354.0	B514.0	A850.0			12–25	½–1	P	2.5–1.8	10–14	245(b)	800(b)
242.0	355.0	520.0	B850.0			25–75	1–3	P	4–2.5	6–10	200(b)	650(b)
295.0	C355.0	535.0				>75	>3	C	8.5	3	170(b)	550(b)
B295.0	356.0	705.0										
308.0	A356.0	707.0										
Hiduminium RR-350												
Die castings												
360.0				40–100	As-cast	<12	<½	P	1.8–1.4	14–18	200(b)	650(b)
A360.0						12–25	½–1	P	2.5–1.8	10–14	185(b)	600(b)
380.0						25–75	1–3	P	4–2.5	6–10	150(b)	500(b)
A380.0						>75	>3	...	...	...	...	...
C443.0				70–125	Solution treated and aged	<12	<½	P	1.8–1.4	14–18	170(b)	550(b)
518.0						12–25	½–1	P	2.5–1.8	10–14	150(b)	500(b)
						25–75	1–3	P	4–2.5	6–10	120(b)	400(b)
						>75	>3	...	...	...	...	...
383.0				40–100	As-cast	<12	<½	P	1.8–1.4	14–18	170(b)	550(b)
A384.0						12–25	½–1	P	2.5–1.8	10–14	150(b)	500(b)
413.0						25–75	1–3	P	4–2.5	6–10	120(b)	400(b)
A413.0						>75	>3	...	...	...	...	...
				70–125	Solution treated and aged	<12	<½	P	1.8–1.4	14–18	150(b)	500(b)
						12–25	½–1	P	2.5–1.8	10–14	135(b)	450(b)
						25–75	1–3	P	4–2.5	6–10	105(b)	350(b)
						>75	>3	...	...	...	...	...
390.0				40–100	As-cast	<12	<½	P	1.8–1.4	14–18	76	250
392.0						12–25	½–1	P	2.5–1.8	10–14	67	220
						25–75	1–3	P	4–2.5	6–10	56	185
						>75	>3	...	...	...	...	...
				70–125	Solution treated and aged	<12	<½	P	1.8–1.4	14–18	60	200
						12–25	½–1	P	2.5–1.8	10–14	55	180
						25–75	1–3	P	4–2.5	6–10	46	150
						>75	>3	...	...	...	...	...

(a) P, precision; C, claw; B, buttress. (b) Recommended speed for high-carbon-steel blade. Source: Ref 5

carbide cutters, mainly because the speeds are usually greater for carbide cutters. Depth of cut (roughing or finishing), particularly in face milling, has some effect on the rate of feed selected. For end milling, the principal factor affecting rate of feed is cutter diameter (Table 28) because small end milling cutters lack rigidity.

Depth of cut is commonly about 6.35 mm (0.250 in.) for roughing and 0.64 (0.025 in.) or less for finishing. If power is available, depth of cut can be several times greater than the above when large amounts of metal are to be removed.

In machining aircraft components, skin milling, which involves deep cuts (sometimes 50 mm, or 2 in., or more), is common practice. Machines used for skin milling are basically planer mills that have evolved into elaborate tracer-controlled machines. The milling is ordinarily done by means of large peripheral-type (slabbing) cutters.

Cutting fluid should be supplied copiously and under pressure to the tool and workpiece. It is important, particularly with carbide cutters, that the cutting fluid be supplied uniformly and consistently to all parts of the cutter in order to prevent overheating and sudden chilling of the carbide tips. Rapid heating and cooling is extremely detrimental to the life of carbide cutters.

For high-speed milling, emulsions of soluble oil and water at a 1:15 ratio are recommended. For low-speed milling, a ratio of 1:30 is recommended.

Kerosene is effective with form cutters. When modified T-slot cutters were used to mill a blade-root slot in aluminum wheels for axial-flow compressors to a tolerance of less than 0.025 mm (0.001 in.), no cutting fluid except kerosene could produce the required accuracy and surface finish.

Table 32 Nominal speeds and feeds for the power hacksawing of aluminum alloys with high-speed tool steel blades

Material	Hardness, HB (500 kg)	Condition	Material thickness, mm	Material thickness, in.	Thread designation: Pitch, mm	Thread designation: Teeth/in.(a)	Speed, strokes/min	Feed: mm/stroke	Feed: in./stroke	Feed: Pressure(b)
Wrought										
EC 2218 5252 6253	30–80	Cold drawn	<6	<¼	4	6	160	0.15	0.006	M
1060 2219 5254 6262			6–18	¼–¾	4	6	160	0.23	0.009	H
1100 2618 5454 6463			18–50	¾–2	6.3	4	160	0.30	0.012	H
1145 3003 5456 6951			>50	>2	6.3	4	160	0.30	0.012	H
1175 3004 5457 7001	75–150	Solution treated and aged	<6	<¼	4	6	140	0.15	0.006	M
1235 3005 5652 7004			6–18	¼–¾	4	6	140	0.23	0.009	H
2011 4032 5657 7005			18–50	¾–2	6.3	4	140	0.30	0.012	H
2014 5005 6053 7039			>50	>2	6.3	4	140	0.30	0.012	H
2017 5050 6061 7049										
2018 5052 6063 7050										
2021 5056 6066 7075										
2024 5083 6070 7079										
2025 5086 6101 7175										
2117 5154 6151 7178										
Cast										
Sand and permanent mold										
A140 319.0 357.0 A712.0	40–100	As-cast	<6	<¼	4	6	140	0.15	0.006	M
201.0 328.0 359.0 D712.0			6–18	¼–¾	4	6	140	0.23	0.009	H
208.0 A332.0 B443.0 713.0			18–50	¾–2	6.3	4	140	0.30	0.012	H
213.0 F332.0 514.0 771.0			>50	>2	6.3	4	140	0.30	0.012	H
222.0 333.0 A514.0 850.0	70–125	Solution treated and aged	<6	<¼	2.5	10	130	0.15	0.006	M
224.0 354.0 B514.0 A850.0			6–18	¼–¾	4	6	130	0.23	0.009	H
242.0 355.0 520.0 B850.0			18–50	¾–2	6.3	4	130	0.23	0.009	H
295.0 C355.0 535.0			>50	>2	6.3	4	130	0.30	0.012	H
B295.0 356.0 705.0										
308.0 A356.0 707.0										
Hiduminium RR-350										
Die castings										
360.0 A380.0	40–100	As-cast	<6	<¼	4	6	140	0.15	0.006	M
A360.0 C443.0			6–18	¼–¾	4	6	140	0.23	0.009	H
380.0 518.0			18–50	¾–2	6.3	4	140	0.30	0.012	H
			>50	>2	6.3	4	140	0.30	0.012	H
	70–125	Solution treated and aged	<6	<¼	2.5	10	130	0.15	0.006	M
			6–18	¼–¾	4	6	130	0.23	0.009	H
			18–50	¾–2	6.3	4	130	0.23	0.009	H
			>50	>2	6.3	4	130	0.30	0.012	H
383.0	40–100	As-cast	<6	<¼	2.5	10	130	0.15	0.006	M
A384.0			6–18	¼–¾	4	6	130	0.23	0.009	H
413.0			18–50	¾–2	6.3	4	130	0.23	0.009	H
A413.0			>50	>2	6.3	4	130	0.30	0.012	H
	70–125	Solution treated and aged	<6	<¼	2.5	10	130	0.15	0.006	M
			6–18	¼–¾	4	6	130	0.23	0.009	H
			18–50	¾–2	6.3	4	130	0.23	0.009	H
			>50	>2	6.3	4	130	0.23	0.009	H
390.0	40–100	As-cast	<6	<¼	2.5	10	125	0.15	0.006	M
392.0			6–18	¼–¾	4	6	125	0.15	0.006	M
			18–50	¾–2	6.3	4	115	0.15	0.006	M
			>50	>2	6.3	4	115	0.15	0.006	M
	70–125	Solution treated and aged	<6	<¼	2.5	10	120	0.075	0.003	L
			6–18	¼–¾	4	6	120	0.15	0.006	M
			18–50	¾–2	4	6	110	0.15	0.006	M
			>50	>2	6.3	4	110	0.15	0.006	M

(a) Conversions per ISO 2336. (b) L, light; M, medium; H, heavy. Source: Ref 5

Form milling is often an economical method for machining complicated shapes in aluminum. The complexity of form that can be built into the design of a cutter is limited, as when the tangent of the curvature approaches 90° to the horizontal plane. As a result, secondary operations are sometimes required to complete a complex form.

Basic types of form cutters have cam-relieved teeth or shaped teeth. Cam-relieved teeth are sharpened by grinding the tooth faces without changing the form. If surface finish is a secondary consideration, straight-tooth cutters are used because they are easier to sharpen. However, if surface finish is important, helical cutters must be used. Although helical cutters can be operated at higher speeds and greater feeds than straight-tooth cutters, they require considerably greater care when being sharpened.

Shape form cutters are made either with integral teeth or blade inserts. These cutters are less expensive than integral-tooth, cam-relieved cutters, but they require more equipment and more skill in grinding.

The feasibility of form milling depends on several variables, including cutter cost, volume of production, design of workpiece, and type of milling machine available. Thus, for small production lots, the cost of a single form cutter must be compared with the cost of several cutters of less complex shape. For form milling, the workpiece must be relatively large, must have no thin sections, and must be capable of being well secured.

Finally, form milling requires high-power equipment that is rugged and in good operating condition. The following example describes an application of form milling and shows the production quantity required to justify a high-cost form cutter.

Example 9: Cost of Standard Versus Form Milling. The contour of the workpiece shown in Fig. 27 could be milled equally well by using five standard cutters for five separate operations or by using one specially designed form cutter that could mill the contour in a single operation. The operations performed by both methods are listed in the table accompanying Fig. 27. Although the cost of the cutters was the same for form and standard milling, the time required for form milling was only about one-third that for standard milling.

Automatic Control. The generally good machining characteristics of aluminum alloys make them especially well adapted to automatic control, by which extremely complex shapes can be milled efficiently. The following example gives details of an application in which automatic control was used.

Example 10: Milling a 758 kg (1670 lb) Forging With Numerical Control. The part shown in Fig. 28 was made from an alloy 7075 blocker forging that weighed 758 kg (1670 lb). Overall measurements of the forging were 3380 × 1730 × 406 mm (133 × 68 × 16 in.), and those of the finished part were 3330 × 1680 × 356 mm (131 × 66 × 14 in.). The weight of the finished part was 178 kg (393 lb), with 579 kg (1277 lb) of metal having been removed in machining.

With the part held in a special milling fixture, rough profile machining was performed in a tape-controlled three-axis profile milling machine. A special boring fixture was used for rough line boring the 140 and 222 mm (5.50 and 8.75 in.) diam trunnion holes. After roughing cuts were completed, approximately 3.05 mm (0.120 in.) of metal remained to be removed by finish machining after the forging was heat treated to the T6 temper. Special fixtures prevented warpage during heat treating and in finish milling. Machining was completed using the same NC machine. Details of the process, machine, and cutter are given with Fig. 28.

Dimensional accuracy is affected by residual stress in the workpiece, induced stress caused by milling, built-up edge on cutters, and dull cutters. Workpieces with complex shapes or variable section thickness, especially in the aged condition, are more difficult to mill to close tolerances. To achieve dimensional stability and to maintain close tolerances, residual stress must be avoided.

Milling of high-silicon alloy 390 (Ref 2) is accomplished with carbide tools at speeds up to 175 m/min (575 sfm) or with polycrystalline diamond tools at speeds up to 790 m/min (2600 sfm). Moderate positive-rake and

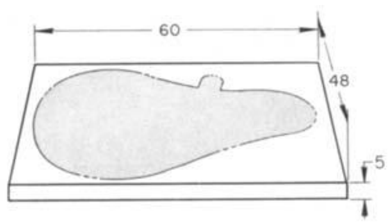


Fig. 32 Shape that was contour band sawed from aluminum plate. Dimensions given in inches

shear angles should be used. Optimum tool life will be achieved if a coolant is used, but alloy 390 can be machined dry with good tool life and with little built-up edge problem.

Compared to other aluminum alloys, alloy 390 has traditionally been milled at rather slow speeds (90 to 175 m/min, or 300 to 575 sfm) using C-2 grade tungsten carbide inserts with a liberal application of flood coolant. Feed rates of 0.10 to 0.30 mm/tooth (0.004 to 0.012 in./tooth) and cutting depths ranging from 0.13 to 5.1 mm (0.005 to 0.20 in.) are common practice.

Cutter geometry is important for ensuring adequate surface finish of the machined casting and a long tool life for the cutting inserts. The entering angle should be 45° to minimize vibration and workpiece frittering. A large positive-rake angle, from 5 to 20°, extends tool life and reduces the tendency toward built-up edge. Clearance angles from 5 to 15° reduce buildup on the edge clearance flank. Pitch depends largely on the diameter of the cutter, depth of cut, and type of cut (roughing or finishing). As with turning, the productivity and tool life performance of the milling operation can be greatly improved through substitution of polycrystalline diamond for carbide.

The first large-volume alloy 390 engine manufactured in the United States was machined on a transfer line designed for gray iron. Instead of broaching, face milling was employed, with an average production tool life of 130 engine blocks per carbide tool.

When compared experimentally with polycrystalline diamond (Fig. 29), the performance of the milling operation was greatly enhanced. At ten times the speed and metal removal rate, tool life would be projected to be in excess of 1000 engine blocks. At the same speed and production rate used for carbide, the diamond tool life would be practically infinite.

This performance improvement was also experienced in a study in which alloy 390 engine blocks were face milled without coolant (Fig. 30). The best tool life achieved when milling with carbide from 100 to 175 m/min (328 to 574 sfm) was approximately 50 engine blocks at lower speed. The feed rate was 0.18 mm/tooth (0.007 in./tooth), and the depth of cut was 1.02 mm (0.040 in.) for roughing and 0.51 mm (0.020 in.) for finishing cuts. The best tool life was achieved using H10 grade cemented carbide. The tool life when dry milling with polycrystalline diamond at speeds ranging from 400 to 800 m/min (1312 to 2625 sfm) was approximately 1000 engine blocks at the lower speed. Surface finish was also greatly improved, indicating that polycrystalline diamond is much less sensitive than carbide to the effect of coolant when milling alloy 390. These results do not indicate that the application of coolant is not desirable to prolong tool life, but it does mean that milling with polycrystalline diamond without coolant, if desired, can be a commercial reality.

Milling of Alloy 380 With Polycrystalline Cutting Tools (Ref 1). An interesting application of diamond tooling is the machining of an aluminum automotive extension housing made of die cast 380 aluminum alloy, which contains approximately 8% Si. The component is about 330 mm (13 in.) long, tapering from about 203 × 254 mm (8 × 10 in.) at one end to a diameter of about 75 mm (3 in.) at the other. The overall shape of the casting is relatively complex, and the nominal wall thickness over most of its area is about 4.0 mm (5/32 in.). A 14-station pallet-type transfer machine performs a number of drilling, tapping, reaming, chamfering, boring, and facing operations on the transmission part.

The large end of the casting requires the machining of a shallow, interruped bore of about 140 mm (5½ in.) in diameter and 9.5 mm (3/8 in.) deep and the facing (milling) of the entire end. To maintain perpendicularity, the boring and the milling tools are combined into a single cutter body. Two of these are used—one for roughing (which also incorporates a tool for chamfering the bore inside diameter), and one for finishing.

The overall configuration of the cutter is essentially a hollow mill with a central boring bar. Tool and workpiece are both shown in Fig. 31. This is fed axially into the part to produce the bore and the chamfer, while the milling teeth rotate beyond the periphery of the part. The cutter is then withdrawn from the bore and fed radially across the surface to be face milled. The milling teeth are positioned farther out axially than the boring and chamfering teeth so that there is no interference.

All cutting edges are polycrystalline diamond. The boring tool is a special diamond-tip 9.5 mm (3/8 in.) inscribed circle triangular insert clamped in place with 2° radial rake and 5° axial rake. The chamfering tool is a 9.5 mm (3/8 in.) square insert with a brazed diamond cutting edge. This insert is presented to the cut with 0° radial and 5° axial rake. Although chamfering is done only at the roughing station, a dummy cartridge is used to maintain balance of the finishing cutter—a necessity at these speeds.

Twelve milling teeth are set on an effective diameter of 356 mm (14 in.). These are

Table 33 Wheel recommendations for grinding aluminum alloys

			Surface grinding										
			Horizontal spindle, reciprocating table					Vertical spindle, rotary table					
												Wheel identification	
Material	Hardness, HB (500 kg)	Condition	Wheel speed, m/s (sfm)	Table speed, m/min (sfm)	Down feed, mm/pass (in./pass)	Cross feed, mm/pass (in./pass)	Wheel identification, ISO (ANSI)(a)	Wheel speed, m/s (sfm)	Table (work) speed, m/min (sfm)	Down feed per revolution of table, mm (in.)	Operation	Narrow work area, ISO (ANSI)	Broad work area, ISO (ANSI)
Wrought													
EC 3005 6066 1060 4032 6070 1100 5005 6101 1145 5050 6151 1175 5052 6253 1235 5056 6262 2011 5083 6463 2014 5086 6951 2017 5154 7001 2018 5252 7004 2021 5254 7005 2024 5454 7039 2025 5456 7049 2117 5457 7050 2218 5652 7075 2219 5657 7079 2618 6053 7175 3003 6061 7178 3004 6063	30–150	Cold drawn or solution treated and aged	20–25 (4000–5000)	15–30 (50–100)	Rough: 0.075 (0.003) Finish: 0.025 max (0.001 max)	1.25–12.5 (0.050–0.500) (Max: 1/3 of wheel width)	C46JV(b) (C46JV)(b)	18–30 (3500–6000)	30–76 (100–250)	0.025–0.100 (0.001–0.004)	Roughing Finishing	CA46IB (CA46IB) CA60IB (CA60IB)	CA46GB (CA46GB) CA60GB (CA60GB)
Cast													
Sand and permanent mold A140 319.0 707.0 201.0 355.0 A712.0 208.0 C355.0 D712.0 213.0 B443.0 713.0 222.0 514.0 771.0 224.0 A514.0 850.0 242.0 B514.0 A850.0 295.0 520.0 B850.0 B295.0 535.0 308.0 705.0 Hiduminium RR-350 **Die castings** C443.0 518.0	40–125	As-cast or solution treated and aged	20–25 (4000–5000)	15–30 (50–100)	Rough: 0.075 (0.003) Finish: 0.025 max (0.001 max)	1.25–12.5 (0.050–0.500) (Max: 1/3 of wheel width)	C46JV(b) (C46JV)(b)	18–30 (3500–6000)	30–76 (100–250)	0.025–0.100 (0.001–0.004)	Roughing Finishing	CA46IB (CA46IB) CA60IB (CA60IB)	CA46GB (CA46GB) CA60GB (CA60GB)
Sand and permanent mold 328.0 333.0 A356.0 A332.0 354.0 357.0 F332.0 356.0 359.0 **Die castings** 360.0 383.0 413.0 A360.0 A384.0 A413.0 380.0 390.0 A380.0 392.0	40–125	As-cast or solution treated and aged	20–25 (4000–5000)	15–30 (50–100)	Rough: 0.075 (0.003) Finish: 0.025 max (0.001 max)	1.25–12.5 (0.050–0.500) (Max: 1/3 of wheel width)	A46IV (A46IV)	18–30 (3500–6000)	30–76 (100–250)	0.025–0.100 (0.001–0.004)	Roughing Finishing	CA46IB (CA46IB) CA60IB (CA60IB)	CA46GB (CA46GB) CA60GB (CA60GB)

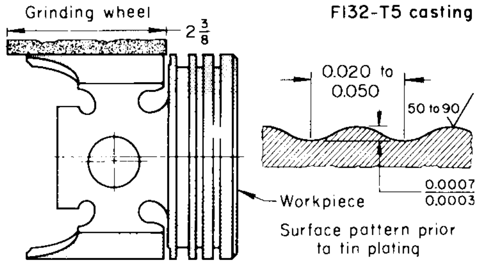
(continued)

(a) Wheel recommendations are for wet grinding. For dry grinding, use a softer-grade wheel. (b) Wax filled. (c) Wheel recommendations are for the wet grinding of 50–100 mm (2–4 in.) diam work. For dry grinding, use a softer-grade wheel. For larger-diameter work, use a softer-grade and/or coarser-grit wheel. For smaller-diameter work, use a harder-grade wheel. Wheel recommendations also apply to plunge grinding applications. (d) As recommended starting conditions, use a regulating wheel angle with a positive inclination of 3° and a regulating wheel speed of 25–40 rev/min. (e) Wheel recommendations are for wet grinding 20–50 mm (0.8–2 in.) diam work. For larger-diameter work, use a softer-grade and/or coarser-grit wheel. For smaller-diameter work, use a harder-grade wheel. (f) Maximum hole length is 2.5 times hole diameter. (g) Maximum wheel width is 1.5 times wheel diameter. (h) Wheel recommendations are for wet grinding 20–50 mm (0.8–2 in.) diam holes. For larger holes, use the same or softer-grade wheel. For smaller holes, use a harder-grade wheel. Source: Ref 5

Table 33 (continued)

Material	Hardness, HB (500 kg)	Condition	Cylindrical grinding					Centerless grinding					Internal grinding(f)(g)				
			Wheel speed, m/s (sfm)	Work speed, m/min (sfm)	Infeed on diameter, mm/pass (in./pass)	Traverse wheel width per revolution of work	Wheel identification, ISO (ANSI)(c)	Wheel speed, m/s (sfm)	Through feed of work, m/min (in./min)(d)	Infeed on diameter, mm/pass (in./pass)	Wheel identification, ISO (ANSI)(e)	Wheel speed, m/s (sfm)	Work speed, m/min (sfm)	Infeed on diameter, mm/pass (in./pass)	Traverse wheel width per revolution of work	Wheel identification, ISO (ANSI)(h)
Wrought																	
EC 2218 5252 6253 1060 2219 5254 6262 1100 2618 5454 6463 1145 3003 5456 6951 1175 3004 5457 7001 1235 3005 5652 7004 2011 4032 5657 7005 2014 5005 6053 7039 2017 5050 6061 7049 2018 5052 6063 7050 2021 5056 6066 7075 2024 5083 6070 7079 2025 5086 6101 7175 2117 5154 6151 7178	30–150	Cold drawn or solution treated and aged	28–33 (5500–6500)	15–46 (50–150)	Rough: 0.050 (0.002) Finish: 0.013 max (0.0005 max)	⅓ ⅙	C54JV(b) (C54JV)(b)	28–33 (5500–6500)	1.3–3.8 (50–150)	Rough: 0.13 (0.005) Finish: 0.038 max (0.0015 max)	C54JV (C54JV)	25–33 (5000–6500)	23–60 (75–200)	Rough: 0.075 (0.003) Finish: 0.005 max (0.0002 max)	⅓ ⅙	C46JV (C46JV)
Cast																	
Sand and permanent mold A140 295.0 514.0 A712.0 201.0 B295.0 A514.0 D712.0 208.0 308.0 B514.0 713.0 213.0 319.0 520.0 771.0 222.0 355.0 535.0 850.0 224.0 C355.0 705.0 A850.0 242.0 B443.0 707.0 B850.0 Hiduminium RR-350 **Die castings** C443.0 518.0	40–125	As-cast or solution treated and aged	28–33 (5500–6500)	15–46 (50–150)	Rough: 0.050 (0.002) Finish: 0.013 max (0.0005 max)	⅓ ⅙	C54JV(b) (C54JV)(b)	28–33 (5500–6500)	1.3–3.8 (50–150)	Rough: 0.13 (0.005) Finish: 0.038 max (0.0015 max)	C54JV (C54JV)	25–33 (5000–6500)	23–60 (75–200)	Rough: 0.075 (0.003) Finish: 0.005 max (0.0002 max)	⅓ ⅙	C46JV (C46JV)
Sand and permanent mold 328.0 333.0 A356.0 A332.0 354.0 357.0 F332.0 356.0 359.0 **Die castings** 360.0 A360.0 390.0 A413.0 A360.0 383.0 392.0 380.0 A384.0 413.0	40–125	As-cast or solution treated and aged	28–33 (5500–6500)	15–46 (50–150)	Rough: 0.050 (0.002) Finish: 0.013 max (0.0005 max)	⅓ ⅙	A54IV (A54IV)	28–33 (5500–6500)	1.3–3.8 (50–150)	Rough: 0.13 (0.005) Finish: 0.038 max (0.0015 max)	A60IV (A60IV)	25–33 (5000–6500)	23–60 (75–200)	Rough: 0.075 (0.003) Finish: 0.005 max (0.0002 max)	⅓ ⅙	A60HV (A60HV)

(a) Wheel recommendations are for wet grinding. For dry grinding, use a softer-grade wheel. (b) Wax filled. (c) Wheel recommendations are for the wet grinding of 50–100 mm (2–4 in.) diam work. For dry grinding, use a softer-grade wheel. For larger-diameter work, use a softer-grade and/or coarser-grit wheel. For smaller-diameter work, use a harder-grade wheel. Wheel recommendations also apply to plunge grinding applications. (d) As recommended starting conditions, use a regulating wheel angle with a positive inclination of 3° and a regulating wheel speed of 25–40 rev/min. (e) Wheel recommendations are for wet grinding 20–50 mm (0.8–2 in.) diam work. For larger-diameter work, use a softer-grade and/or coarser-grit wheel. For smaller-diameter work, use a harder-grade wheel. (f) Maximum hole length is 2.5 times hole diameter. (g) Maximum wheel width is 1.5 times wheel diameter. (h) Wheel recommendations are for wet grinding 20–50 mm (0.8–2 in.) diam holes. For larger holes, use the same or softer-grade wheel. For smaller holes, use a harder-grade wheel. Source: Ref 5



Machine	Cylindrical grinder, 152 × 457 mm (6 × 18 in.), with automatic infeed electric-cycle control and semiautomatic profile dresser
Wheel	
Type	A-46-K5-V
Size, mm (in.)	508 × 60 × 305 (20 × 2⅜ × 12)
Life per dressing	100 pieces
Operating conditions	
Wheel speed, at 1300 rev/min, m/min (sfm)	2100 (6800)
Work speed, rev/min	96
Feed/pass, mm (in.)	0.41–0.46 (0.016–0.018)
Grinding fluid	Soluble oil:water (1:25)
Production rate, pieces/h	106
Time/pass, s(a)	20
Surface finish, μm (μin.)	1.25–2.25 (50–90)

(a) Infeed set with automatic timer

Fig. 33 Permanent mold cast piston (90 to 120 HB) that was ground in a single pass, removing 0.41 to 0.46 mm (0.016 to 0.018 in.) of stock. Dimensions in figure given in inches

Table 34 Abrasive cutoff wheels for cutting aluminum alloys

Material				Hardness, HB (500 kg)	Condition	Wheel identification: Dry cutting ISO	Dry cutting ANSI	Wet cutting ISO	Wet cutting ANSI
Wrought									
EC	2218	5252	6253	30–150	Cold drawn or solution treated and aged	A36—RB	A36—RB	A46—PR	A46—PR
1060	2219	5254	6262						
1100	2618	5454	6463						
1145	3003	5456	6951						
1175	3004	5457	7001						
1235	3005	5652	7004						
2011	4032	5657	7005						
2014	5005	6053	7039						
2017	5050	6061	7049						
2018	5052	6063	7050						
2021	5056	6066	7075						
2024	5083	6070	7079						
2025	5086	6101	7175						
2117	5154	6151	7178						
Cast									
Sand and permanent mold									
A140	295.0	514.0	A712.0	40–125	As-cast or solution treated and aged	A36—RB	A36—RB	A46—PR	A46—PR
201.0	B295.0	A514.0	D712.0						
208.0	308.0	B514.0	713.0						
213.0	319.0	520.0	771.0						
222.0	355.0	535.0	850.0						
224.0	C355.0	705.0	A850.0						
242.0	B443.0	707.0	B850.0						
Hiduminium RR-350									
Die castings									
C443.0	518.0								
Sand and permanent mold									
328.0	333.0	A356.0		40–125	As-cast or solution treated and aged	A36—RB	A36—RB	A46—PR	A46—PR
A332.0	354.0	357.0							
F332.0	356.0	359.0							
Die castings									
360.0	383.0	413.0							
A360.0	A384.0	A413.0							
380.0	390.0								
A380.0	392.0								

Source: Ref 5

cartridge-type cutters with brazed diamond tips having the following geometry: 15° lead angle (the inner edge), 5° radial rake, 5° axial rake, 6° clearance, and 1.52 mm (0.060 in.) nose radius. On the finishing cutter, one of these is ground with a 0.76 mm (0.030 in.) wiper flat instead of the radius, which was found to improve significantly the surface finish produced on the part.

These combination cutters are run at 3450 rev/min for both roughing and finishing, yielding a cutting speed of 1514 m/min (4967 sfm) for the boring and chamfering operations and 3854 m/min (12 645 sfm) for the face milling operations. In roughing, material removal is approximately 2.03 mm (0.080 in.) in the bore and 1.65 mm (0.065 in.) on the face. In finishing, 0.38 mm (0.015 in.) is removed from both surfaces.

Sawing

The contour cutting of aluminum alloys is usually done by band sawing; straight cutting is done on a circular saw, band saw, power hacksaw, or abrasive cutoff wheel. Circular saws or band saws are preferred for the rapid cutoff of rod and bar stock; either can be readily adapted for high-speed, automatic work handling.

Linear feed rates as high as 762 to 2030 mm/min (30 to 80 in./min) are sometimes used in the cutoff of stock 50 to 203 mm (2 to 8 in.) thick, for cutting rates of 0.032 to 0.16 m^2/min (50 to 250 in.^{2}/min). Recommended tooth angles and contours for the three types of saws are given in Tables 29 to 32.

Circular Sawing. Peripheral speeds are 45 to 4600 m/min (150 to 15 000 sfm) for high-speed steel and carbide-tip circular saws, depending on the saw material, the type of cut, and the ability of the machine to withstand high speeds (Tables 29 and 30). The limiting factor is usually the maximum safe operating speed of the machine and blade.

The feed rate for circular sawing ranges from 0.05 to 7.6 mm/tooth (0.002 to 0.30 in./tooth), depending mainly on the alloy being sawed. The width of the section being sawed and the speed of the saw have some bearing on the rate of feed.

Band Sawing. Hard-tooth, flexible-back carbon steel bands are used at 46 to 305 m/min (150 to 1000 sfm) on cast and wrought aluminum alloys (Table 31). The following example compares milling and contour band sawing for producing a shape from an aluminum alloy plate. In this application, band sawing was faster and less expensive.

Example 11: Contour Sawing of a Large Plate. An aluminum alloy plate weighing 653 kg (1440 lb) and measuring 1220 × 1525 × 127 mm (48 × 60 × 5 in.) was rough machined in a milling machine to obtain the contour shown in Fig. 32. Milling required about 4½ h. The same part was produced in 67 min in a contour band saw, cutting at a rate of 9700 mm^2/min (15 in.^{2}/min).

Sawing was done with a high-speed steel blade (hook tooth; raker set; 6 pitch; 25 mm, or 1 in., wide; 0.89 mm, or 0.035 in., thick), using a heavy-duty chemical solution as a cutting fluid. A dimensional tolerance of ±0.38 mm (±0.015 in.) was maintained.

Power hacksawing is done at 110 to 160 strokes per min with a feed of 0.15 to 0.30 mm/stroke (0.006 to 0.012 in./stroke) (Table 32). The higher speed is recommended for sawing nonheat-treated cast aluminum alloys, and the lower speed (110 strokes per min) for abrasive materials such as 390 and 392 solution-treated and aged die cast alloys.

Cutting Fluid. Although circular saws can be operated satisfactorily on aluminum alloys at moderate speeds and medium cuts

Table 35 Nominal speeds for the abrasive belt grinding of aluminum alloys

Material	Hardness, HB (500 kg)	Condition	Operation	Abrasive type	Grain size	Belt speed(a) m/s	Belt speed(a) sfm	Contact wheel Type(b)	Contact wheel Hardness, Durometer
Wrought and cast	30–150	Cold drawn, as-cast, or solution treated and aged	Roughing	Al_2O_3, SiC	24–80	25–33	5000–6500	SR, SFR	70–95
			Polishing	Al_2O_3, SiC	100–240	25–40	5000–8000	SR, SFR, B	20–60

(a) Use lower values of belt speed when good surface integrity is required. (b) SR, serrated rubber; SFR, smooth face rubber; B, buff type. Serrations are usually at a 45° angle, although some heavy stock removal operations use a 60° angle. Widths of lands and grooves vary from narrow lands and wide grooves for fast aggressive cuts to wide lands and narrow grooves for intermediate and finishing operations, depending on workpiece shape and operating conditions. Source: Ref 5

Table 36 Honing recommendations for aluminum alloys

Material	Hardness, HB (500 kg)	Honing stone material Type	Honing stone material Grade, ANSI or ISO	Grain size at a surface roughness, R_a, μm (μin.), of: 0.025–0.125 (1–5)	0.15–0.25 (6–10)	0.30–0.50 (11–20)	0.53–0.75 (21–30)	>0.75 (>30)	Spindle motion, m/min (sfm) Rotating speed	Spindle motion, m/min (sfm) Reciprocating speed	Working pressure, kPa (psi)	Cutting fluid
Wrought and cast	30–450	Silicon carbide or diamond (metal bonded)	L(a)	600 (600)	500 (500)	400 (400)	280 (280)	220 (220)	15.2–64 (50–210)	2.7–22.9 (9–75)	276 (40)	70–30 kerosene/ oil (sulfurized or chlorinated)

(a) L, a medium grade. Source: Ref 5

without a cutting fluid, it is advisable to supply copious amounts of soluble-oil:water emulsion (1:20) for all high-speed cutting. The cutting fluid should flood the blade and workpiece under slight pressure and it should be filtered or settled before recycling. In some applications, the addition of a little kerosene or lard oil to the emulsion has been beneficial. Soap solutions can be substituted for oil emulsions.

For band saws, some cutting fluid is essential for all but the lightest cuts. A wide selection of compounds is available, ranging from tallow or grease sticks to kerosene-thinned mineral-base lubricating oil or emulsions of soluble oil and water. It is often more convenient to use a fluid lubricant, supplied generously through a recycling system. For hacksawing, procedures are similar to those for band sawing.

Grinding

The harder, free-cutting aluminum alloys are comparatively easy to grind. The non-free-cutting alloys, particularly in their softer tempers, are likely to clog grinding wheels, and they do not finish to as bright and smooth a surface as the harder alloys.

Abrasive Wheels. For grinding aluminum alloys, a silicon carbide abrasive in a flexible base is generally preferred. Aluminum oxide is seldom recommended, except for piston grinding and in cutoff wheels. Wheels of medium hardness, about 46-grit size, and with a synthetic resin bond work best for roughing. For finishing, wheels of finer grit size (to about 60) and with a vitrified bond are generally used. Recommendations for wheels for several different types of grinding operations are given in Table 33 (the ANSI system for identifying the characteristics of grinding wheels is explained in the articles "Grinding Equipment and Processes" and "Superabrasives" in this Volume). The following example describes specific grinding practice for pistons.

Example 12: Grinding of Pistons. After extensive testing, an A-46-K5-V wheel was selected for grinding the skirt of permanent mold cast F132-T5 pistons (Fig. 33). Requirements for the wheel were:

- Satisfactory performance without frequent redressing
- Ability to hold the required profile
- Production of a 1.25 to 2.25 μm (50 to 90 μin.) finish

The A-46-K5-V wheel was capable of grinding 100 parts between redressings and met the other requirements.

A special hydraulic dresser, equipped with a profile bar, was used to dress the wheel. The dressing tool was a ¾-carat diamond that was turned in its holder after eight dressings to maintain a sharp point. Details of this grinding operation are given in the table accompanying Fig. 33.

Cutoff Wheels. Abrasive cutoff wheels for aluminum are usually of aluminum oxide, as indicated in Table 34. For wet grinding, which is generally recommended, the bond is usually rubber. For dry grinding, either rubber or resinoid bond can be used. Typical specifications are A-20-U6-R (aluminum oxide, 20 grain size, U grade, 6 structure, rubber bond) and A-24-S7-B (aluminum oxide, 24 grain size, S grade, 7 structure, resinoid bond). The width of the wheels ranges from 0.08 to 3.96 mm (0.003 to 0.156 in.); diameter varies from 25 to 762 mm (1 to 30 in.). These wheels cut to an accuracy of a few thousandths of an inch.

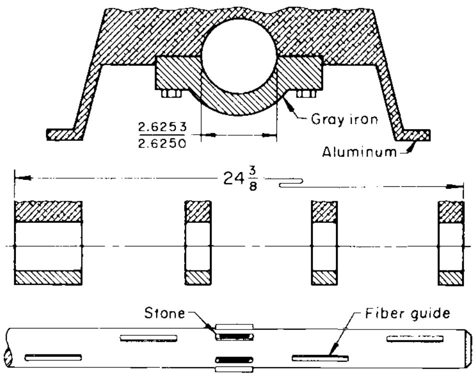
Reinforced (Flexible) Wheels. In cutting off gates and sprues from castings of irregular shape, excessive wheel breakage can occur because of very high pressure on the sides of the wheel. To eliminate this problem and to provide for safe operation, special reinforced, flexible, abrasive wheels have been developed. These wheels are manufactured from laminated sheets of cotton fiber filled with abrasive grain and are used for operations ranging from heavy grinding to light sanding: cutting off, sharpening, deburring, and finishing. Such wheels used for aluminum are C-24-O14-B to C-24-U14-B.

Speed. Typical wheel speeds for several types of grinding are given in Table 33. Sometimes, by adjusting the speed, a wheel that was previously unsuited for a particular operation can be made to grind satisfactorily. For example, if a wheel is too soft, increasing the speed will give a harder action.

Using the recommended speed for a wheel is important, not only from the standpoint of grinding results but also to ensure safety. Stress from centrifugal force increases greatly as wheel velocity increases. The force tending to pull a wheel apart will be four times greater at 3600 rev/min than at 1800 rev/min.

Grinding Fluid. Neutral soluble-oil emulsions are satisfactory for grinding aluminum. The addition of a wetting agent (detergent) is sometimes helpful. Although an emulsion of 1 part soluble oil to 35 parts water is commonly used, a better cushioning effect, which also prevents clogging of the wheel, is obtained by using more oil and less water. One manufacturer, in grinding soft aluminum alloy castings, went progressively from a mixture of 1 part oil to 20 parts water to a mixture of 1 part oil to 6 parts water before obtaining satisfactory results.

In rough grinding, where stock removal is the primary objective, a generous application of wax or stick grease is often satisfactory. This type of lubricant is often used



Speed, at 900 rev/min, m/min (sfm)	190 (620)
Spindle reciprocation rate, strokes/min	60
Hone speed, m/min (sfm)	21 (70)
Stock removed, mm (in.)	0.038–0.51 (0.0015–0.020)
Honing time, s	25
Cutting fluid	Mineral seal oil
Size control method	Air gage
Stone life, per set (average)	300 assemblies
Production rate, pieces/h	75
Bore alignment before honing, mm (in.)	0.05 (0.002)
Bore alignment after honing, mm (in.)	0.018 (0.0007)
Specified tolerance, mm (in.)	±0.008 (±0.0003)
Specified finish, μm (μin.)	0.50–0.63 (20–25)

Fig. 34 Die cast engine block that was honed with gray iron bearing caps in place, and honing tool that was used (bottom). Dimensions in figure given in inches

when the application of liquids is not feasible.

Belt Lubricant. Grinding fluids or stick lubricants improve the finish produced by a coated abrasive belt and prevent glazing of the belt. Lubricants used range from water to stick waxes; standard cutting oils and soluble-oil mixtures are often used in high-production belt machines.

Greases cushion the penetrating action of the abrasive into the work and produce finer finishes than are obtained in dry grinding. Although the crest of each grain is free to cut, the grease prevents deep scratches.

Greases are used for the offhand grinding of aluminum die castings. When stock removal is the primary objective, a light grease should be used, preventing the belt from loading but permitting a maximum amount of abrasive penetration into the metal being ground. For a good finish, a heavy grease should be used.

Some operations use two types of grease, side by side, on the same abrasive belt. One side is used for a high rate of stock removal, and the other for a finer finish. If the belt becomes loaded despite the use of grease, an application of kerosene will free the belt of embedded particles.

Liquid lubricants are primarily used to prevent belt glazing. When water is used, a rust inhibitor should be added.

Soluble oil mixed with water is effective in grinding aluminum if stock removal is the primary objective. Table 35 lists data for abrasive belt grinding.

Honing

Aluminum alloys are honed by methods similar to those used for other metals (see the article "Honing" in this Volume). Resin bond abrasives are preferred; sulfurized mineral-base oil or lard oil mixed with kerosene is used to flush the abrasive sticks clean and to carry away heat. Table 36 lists honing specifications for aluminum alloys.

Honing is used to finish anodized aluminum surfaces (primarily the bores of some small aluminum engine blocks) and most aircraft hydraulic cylinders. Long anodized aluminum tubes that are components of in-flight refueling apparatuses are finished by manual honing. The tubes are chucked in a lathe, and the honing tool is moved manually. The oil supply is attached to the honing tool in such a manner that the flow of oil is directed where most needed. This method of honing is also used for finishing connecting-rod journals or crankshafts in aircraft overhaul shops.

The following example describes the technique and conditions employed in the honing of main bearing bores in aluminum engine blocks. The aluminum blocks and the gray iron bearing caps were honed simultaneously with a silicon carbide abrasive, even though, if honed separately, gray iron and aluminum would be honed with different abrasives.

Example 13: Simultaneous Honing of Aluminum and Gray Iron. Main-bearing bores in six-cylinder, die cast aluminum engine blocks were honed with gray iron bearing caps bolted in place, as shown in Fig. 34. The blocks were fixtured for honing and were held in position by a light clamping force on the head faces. Each block was held securely so that it could resist torsional forces from the honing tool, but adjustment of clamping force was critical because excessive force would distort the block. Hydraulic cone expanders automatically fed out the single bank of six carbon-bonded silicon carbide stones (50 × 6.4 mm, or 2 × ¼ in., 120 grit). Fifteen fiber guides, each 75 × 6.4 mm (3 × ¼ in.), were incorporated in the honing tool. Processing details are given in the table with Fig. 34.

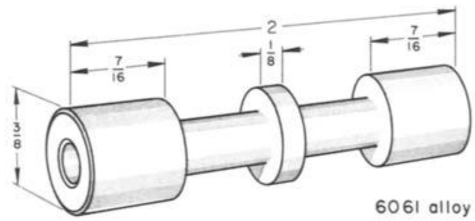


Fig. 35 Part from which 0.005 mm (0.0002 in.) of metal was removed by lapping to smooth the surface to 0.025 μm (1 μin.). Dimensions given in inches

Lapping

The methods used for lapping aluminum are the same as those used for other metals (see the article "Lapping" in this Volume). However, because similar finishes often can be produced by other methods at less cost, lapping is seldom used. In the following example, an extremely smooth finish of 0.025 μm (1 μin.) on an anodized part was produced by two methods of lapping.

Example 14: Lapping Hard-Anodized Alloy 6061. The hard-anodized aluminum part (surface hardness equivalent to 65 HRC) shown in Fig. 35 required removal of 0.005 mm (0.0002 in.) from the three lands to produce a finish of 0.025 μm (1 μin.). Diamond abrasive (8000 mesh) in a paste vehicle was used for both centerless roll lapping and lapping in a two-plate machine. The roll lapper, at a rotation speed of 100 rev/min (large roll) and a stroke speed of 50 mm/min (2 in./min), produced ten parts per hour. The two-plate machine, which had upper and lower cast iron laps 406 mm (16

Table 37 Comparison of data and characteristics of systems for the chemical milling of titanium, steel, aluminum, and nickel- and cobalt-base alloys

Metal	Principal etchant	Etch rate, mm/min	Etch rate, in./min	Optimum etch depth, mm	Optimum etch depth, in.	Etchant temperature, °C	Etchant temperature, °F	Average surface roughness, R_a, μm	Average surface roughness, R_a, μin.
Titanium alloys	Hydrofluoric acid	0.015–0.030	0.0006–0.0012	3.2	0.125	46±2.7	115±5	0.4–2.5	16–100
Steels	Hydrochloric acid-nitric acid	0.015–0.030	0.0006–0.0012	3.2	0.125	63±2.7	145±5	0.8–3.2	30–120
Aluminum alloys	Sodium hydroxide	0.020–0.030	0.0008–0.0012	3.2	0.125	90±2.7	195±5	2.0–3.2	80–120
Nickel- and cobalt-base alloys	Nitric acid-hydrochloric acid-ferric chloride	0.010–0.038	0.0004–0.0015	3.2	0.125	60±2.7	140±5	1.0–3.8	40–150

Source: Ref 5

Table 38 Properties of maskant materials exposed to acids or alkalies that are widely used in the chemical milling of aluminum alloys

Property	Maskant material: Butyl rubber	Acrylonitrile rubber	Neoprene rubber
Ease of manufacture	Fair	Fair	Good
Shelf life, months	4–6	3–6	6–8
Solids, %	20–25	15–25	25–35
Ease of application			
Dipping	Good	Good	Good
Flow coating	Good	Good	Good
Air spraying	Poor	Poor	Fair
Type of cure	Heat	Air or heat	Air or heat
Tensile strength, MPa (ksi)			
Air-dried, 24 h	...	7 (1.0) max	14 (2.0) max
Heat cured	6–11 (0.87–1.6)	11–17.5 (1.6–2.5)	11–21 (1.6–3.0)
Resistance to etchant			
Deterioration	Very good	Very good	Very good
Permeability	Excellent	Very good	Very good
Heat limit, °C (°F)(a)	150 (300)	120 (250)	95 (200)

(a) Maximum useful temperature for intermittent exposure and for curing temperature. Source: Ref 5

in.) in diameter and 75 mm (3 in.) thick, lapped 1000 parts per hour.

Chemical Milling

Parts can be chemically milled over their entire surface, or the surface can be selectively machined by masking areas that do not require machining. The process is mainly used for parts having large surface areas requiring small amounts of metal removal (see the article "Chemical Milling" in this Volume).

Chemical milling is primarily used for parts having shallow cavities or pockets or requiring overall weight reduction. The main application has been in the aerospace industry to obtain maximum strength-to-weight ratios, but the process has also been receiving attention from other industries. Extremely large parts such as aircraft skin and fuselage sections and airframe extrusions are chemically milled.

Table 37 compares the chemical milling of aluminum to that of other metals typically machined by this process. In contrast to titanium alloys, steels, and nickel- and cobalt-base alloys, which use acid as an etchant, caustic soda is the usual etchant for aluminum alloys.

When selective machining is involved and the scribe-and-peel method is used, masking material is applied to the workpiece by spraying, dipping, or brushing. The dry maskant is scribed from a pattern and peeled from the workpiece to expose the areas that are to be chemically milled. Maskant can also be selectively applied by the silk-screening process or by photographic methods. Properties of maskants used to chemically mill aluminum alloys are listed in Table 38.

Machining takes place when the workpiece, contained in a work basket or held by a rack, is immersed in the chemical solution. Agitation of the work or circulation of the solution is necessary to ensure uniform rates of metal removal from all exposed surfaces. The maskant material is removed from the workpiece by hand or with the aid of a demasking solution.

When scribe-and-peel maskants are used, cuts as deep as 13 mm (0.5 in.) can be made. With the thinner and less chemical-resistant silk-screened masks, the depth of cut is limited to 1.5 mm (0.060 in.), but a more accurate and detailed cavity is possible. Photoresists provide still more detail and accuracy, but the cut is limited to a depth of 1.3 mm (0.050 in.). Sheets and plates are tapered in thickness by immersing and withdrawing the workpiece from the etching bath at a controlled rate. Steps are produced by repeated cycles of etching with different areas masked. The depth of cut is limited to about 13 mm (½ in.) in plate materials and to less than this with forgings, castings, and extrusions.

Table 39 lists data for the surface roughness of three forms of aluminum alloys that were subjected to the chemical milling process. The effect of depth of cut on the surface finish of 7075 aluminum sheet is shown in Fig. 36. Typical tolerances of various depths of cut are:

Depth of cut, mm	Depth of cut, in.	Tolerance, mm	Tolerance, in.
0–1.25	0–0.050	0.025	0.001
1.25–2.50	0.051–0.100	0.038	0.0015
2.5–6.4	0.101–0.250	0.05	0.002
6.4–12.7	0.251–0.500	0.075	0.003

Additional information is available in the article "Chemical Milling" in this Volume. The following example illustrates the use of chemical milling to reduce the weight of a 7072 aluminum aircraft part.

Example 15: Fuselage Section Made From Preformed 7072 Aluminum Alloy Plate. The fuselage skin section shown in Fig. 37 was chemically milled after stretch forming. The web areas comprising about 80% of the surface area (one side) of the originally 6.4 mm (¼ in.) thick section were etched to a thickness of 0.64 mm (0.025 in.),

Table 39 Surface roughness achieved by the chemical milling of aluminum alloys

Form	Surface roughness, R_a, after 0.25–0.40 mm (0.010–0.015 in.) removed, μm	μin.
Sheet	2.00–3.8	80–150
Casting	3.8–7.6	150–300
Forging	2.50–6.3	100–250

Source: Ref 5

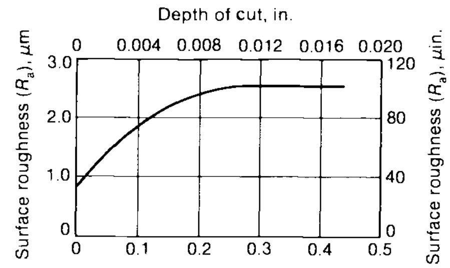


Fig. 36 Relationship between surface finish and depth of cut for the chemical milling of 7075 aluminum sheet. Source: Ref 5

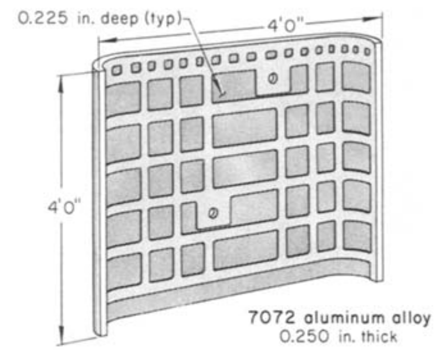
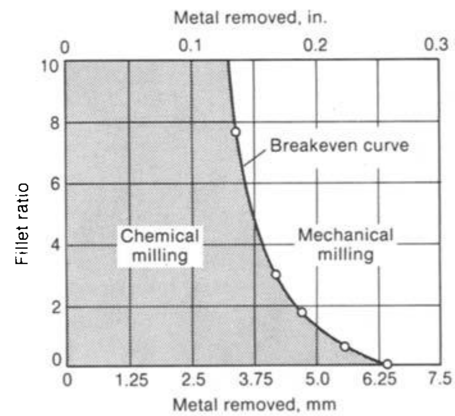


Fig. 37 Preformed aircraft skin section chemically contoured to reduce weight. Dimensions given in inches



$$\text{Fillet ratio} = \frac{\text{total length of fillet (ft)}}{\text{total area of part (ft}^2)}$$

Fig. 38 Breakeven between chemical and mechanical milling. Choice between them depends on fillet ratio and depth of cut.

reducing the weight of the part from 27 to 6.8 kg (60 to 15 lb).

Before the development of chemical milling, parts of this type were made by welding or riveting reinforcing sheet sections to a formed 0.64 mm (0.025 in.) thick skin; there was no practical method of forming such a panel after machining or of machining such a panel after forming. Chemical milling lowered manufacturing cost by greatly reducing the number of parts and eliminating assembly time.

Chemical Versus Mechanical Milling. Figure 38 presents a typical curve for choosing the most economical of these two methods for removing metal from flat parts on which large areas having complex or wavy peripheral outlines are to be reduced in thickness. Fillet ratio and thickness of metal to be removed are used as the basis for evaluation. For this specific application, metal thicknesses greater than 6.35 mm (0.250 in.) should be removed mechanically; thicknesses less than 3.18 mm (0.125 in.), chemically. Between these two values, the choice depends on fillet ratio, which governs the weight penalty.

ACKNOWLEDGMENT

ASM INTERNATIONAL would like to thank Alan McMechan, McDonnell Douglas Canada; Eli Levy, The de Havilland Aircraft Company of Canada; and R.A. Morley, Reynolds Aluminum, for their assistance in updating this article. Mr. McMechan prepared the section "Soft Spots in Machined Aluminum Parts," Mr. Morley supplied data on the machining of high-silicon aluminum alloys, and Mr. Levy provided information on machining practices in the aerospace industry.

REFERENCES

1. Machining Aluminum, Special Report 702, *Am. Mach.*, March 1978
2. J.C. Miller, "Machining High Silicon Aluminum," Paper G-T81-035, presented at the 11th International Die Casting Congress and Exposition, Cleveland, OH, Society of Die Casting Engineers, June 1981
3. A. McMechan, "Soft Spots in Machined Aluminum Parts," Paper presented at the Western Metal & Tool Exposition & Conference, Los Angeles, CA, McDonnell Douglas Corporation, March 1982
4. *Machining Alcoa Aluminum*, Aluminum Company of America, 1967
5. *Machining Data Handbook*, Vol 1 and 2, 3rd ed., Metcut Research Associates, Inc., 1980