MACHINING THE AUSTENITIC CHROMIUM-NICKEL STAINLESS STEELS

A PRACTICAL GUIDE TO THE USE OF NICKEL-CONTAINING ALLOYS

Nº 1130

Produced by INCO

Distributed by NICKEL INSTITUTE

Nickel INSTITUTE

knowledge for a brighter future

MACHINING THE AUSTENITIC CHROMIUM-NICKEL STAINLESS STEELS

A PRACTICAL GUIDE TO THE USE
OF NICKEL-CONTAINING ALLOYS
Nº 1130

Originally, this handbook was published in 1972 by INCO, The International Nickel Company Inc. Today this company is part of Vale S.A.
The Nickel Institute republished the handbook in 2022. Despite the age of this publication the information herein is considered to be generally valid.

Nickel Institute

communications@nickelinstitute.org
www.nickelinstitute.org

AISI and ACI Standard Composition Ranges for Wrought and Cast Chromium-Nickel Stainless Steels

American Iron and Steel Institute Classification of Chromium-Nickel Stainless Steels

AISI Type	Composition, %								
	C max	Mn max	P max	S max	Si max	Cr	Ni	Mo	Other
201	0.15	5.50–7.50	.060	.030	1.00	16.00–18.00	3.50–5.50	—	N 0.25 max
202	0.15	7.50–10.00	.060	.030	1.00	17.00–19.00	4.00–6.00	—	N 0.25 max
301	0.15	2.00	.045	.030	1.00	16.00–18.00	6.00–8.00	—	—
302	0.15	2.00	.045	.030	1.00	17.00–19.00	8.00–10.00	—	—
302B	0.15	2.00	.045	.030	2.00–3.00	17.00–19.00	8.00–10.00	—	—
303	0.15	2.00	0.20	0.15 min	1.00	17.00–19.00	8.00–10.00	0.60 max	—
303Se	0.15	2.00	0.20	.06	1.00	17.00–19.00	8.00–10.00	—	Se 0.15 min
304	.08	2.00	.045	.030	1.00	18.00–20.00	8.00–12.00	—	—
304L	.03	2.00	.045	.030	1.00	18.00–20.00	8.00–12.00	—	—
305	0.12	2.00	.045	.030	1.00	17.00–19.00	10.00–13.00	—	—
308	.08	2.00	.045	.030	1.00	19.00–21.00	10.00–12.00	—	—
309	0.20	2.00	.045	.030	1.00	22.00–24.00	12.00–15.00	—	—
309S	.08	2.00	.045	.030	1.00	22.00–24.00	12.00–15.00	—	—
310	0.25	2.00	.045	.030	1.50	24.00–26.00	19.00–22.00	—	—
310S	.08	2.00	.045	.030	1.50	24.00–26.00	19.00–22.00	—	—
314	0.25	2.00	.045	.030	1.50–3.00	23.00–26.00	19.00–22.00	—	—
316	.08	2.00	.045	.030	1.00	16.00–18.00	10.00–14.00	2.00–3.00	—
316L	.03	2.00	.045	.030	1.00	16.00–18.00	10.00–14.00	2.00–3.00	—
317	.08	2.00	.045	.030	1.00	18.00–20.00	11.00–15.00	3.00–4.00	—
D319	.07	2.00	.045	.030	1.00	17.50–19.50	11.00–15.00	2.25–3.00	—
321	.08	2.00	.045	.030	1.00	17.00–19.00	9.00–12.00	—	Ti 5 x C min
347	.08	2.00	.045	.030	1.00	17.00–19.00	9.00–13.00	—	Cb-Ta 10 x C min
348	.08	2.00	.045	.030	1.00	17.00–19.00	9.00–13.00	—	Cb-Ta 10 x C min; Ta 0.10 max; Co 0.20 max
384	.08	2.00	.045	.030	1.00	15.00–17.00	17.00–19.00	—	—
385	.08	2.00	.045	.030	1.00	11.50–13.50	14.00–16.00	—	—

Alloy Casting Institute (SFSA) Classification of Chromium-Nickel Stainless Steel Castings

Cast Alloy Designation	Wrought Alloy Type[1]	Composition, %								
		C max	Mn max	P max	S max	Si max	Cr	Ni	Mo	Other
CD-4MCu	—	.040	1.00	.04	.04	1.00	25–27	4.75–6.00	1.75–2.25	Cu 2.75–3.25
CE-30	—	0.30	1.50	.04	.04	2.00	26–30	8–11	—	—
CF-3	304L	.03	1.50	.04	.04	2.00	17–21	8–12	—	—
CF-8	304	.08	1.50	.04	.04	2.00	18–21	8–11	—	—
CF-20	302	0.20	1.50	.04	.04	2.00	18–21	8–11	—	—
CF-3M	316L	.03	1.50	.04	.04	1.50	17–21	9–13	2.0–3.0	—
CF-8M	316	.08	1.50	.04	.04	1.50	18–21	9–12	2.0–3.0	—
CF-12M	316	0.12	1.50	.04	.04	1.50	18–21	9–12	2.0–3.0	—
CF-8C	347	.08	1.50	.04	.04	2.00	18–21	9–12	—	Cb 8 x C min, 1.0 max or Cb-Ta 10 x C min, 1.35 max
CF-16F	303	0.16	1.50	0.17	.04	2.00	18–21	9–12	1.5 max	Se 0.20–0.35
CG-8M	317	.08	1.50	.04	.04	1.50	18–21	9–13	3.0–4.0	—
CH-20	309	0.20	1.50	.04	.04	2.00	22–26	12–15	—	—
CK-20	310	0.20	1.50	.04	.04	2.00	23–27	19–22	—	—
CN-7M	—	.07	1.50	.04	.04	1.50	18–22	27.5–30.5	2.0–3.0	Cu 3–4

[1]Wrought alloy type numbers are included only for the convenience of those who wish to determine corresponding wrought and cast grades. The chemical composition ranges of the wrought materials differ from those of the cast grades.

Machining the Austenitic Chromium-Nickel Stainless Steels

INTRODUCTION

The several grades of wrought nickel-containing, austenitic stainless steels discussed herein might be grouped as follows in descending order of machinability:

Group 1	AISI 303, 303Se	Free Machining
Group 2	AISI 201, 202, 301, 302, 304, 304L, 321, 347	Non-Free Machining
Group 3	AISI 309, 309S, 310, 316, 316L	Non-Free Machining

Unless stated otherwise, the alloys discussed have been cold drawn to 150-250 Brinell hardness prior to machining.* Lower machining rates would be required for material of higher hardness.

Steels in the annealed condition, especially the non-free machining grades, machine with a thick, gummy chip developing high cutting temperatures at the chip-tool interface. This permits metal to build up on the face of the cutting tool, impairing separation of the chip from the workpiece, the finish of the machined surface and tool life. When the tooling permits, chip breakers or curlers are recommended for machining annealed material to effect improved chip flow and disposal.

Cold drawn material machines with a relatively clean, thin chip that can be controlled without impairing the finish of the workpiece by conventional chip curlers or breakers.

When machined in the cold drawn condition, the free machining alloys with additives such as sulfur or selenium afford chips superior to all of the alternate grades without reservation as to the condition in which they are machined. In some instances the producers of stainless steel have made improvements on the free machining alloys. Seeking their guidance on these improvements would be helpful.

The suggestions noted below facilitate economic machining on a production basis:

a) Use rigid machines with true running spindles.

b) Utilize machines at not more than 75 per cent of rated capacity.

c) Use well mounted, sturdy tools with as little overhang as possible on tool holders.

d) The workpiece should be firmly held and, when possible, backed by a roller rest.

e) Make a turning breakdown cut on hexagonal and like sections before forming.

f) Use a steady, positive feed. Tools should never be allowed to ride the material because this will result in hardening of the workpiece and premature tool dulling or breakage.

g) Grind tools to smooth keen cutting edges.

h) Regrind tools at predetermined intervals.

i) Supply clean, force fed cutting fluid to all machining operations except friction sawing.

TOOL MATERIALS

General purpose high speed steel, high speed steel containing cobalt, cast alloys and cemented carbides are used for machining nickel-containing austenitic stainless steels.

The premium grade high speed steels containing cobalt are often used in preference to the general purpose grades, especially for continuous cuts, because of their superior abrasion resistance and higher hot hardness. These characteristics are also found in cast alloy tools. Under favorable operating conditions the cobalt-containing steels and the cast alloys may be operated at surface cutting speeds about 20 per cent above the general purpose steels. They are not as shock resistant as the general purpose steels and, consequently, due consideration should be given to the type of machining operation, depth of cut, feed and rigidity of machine and work setup.

Cemented carbide tools are operated at several times the surface cutting speeds of the general purpose high speed steels. Usually Grade C-2 cemented carbide is satisfactory. The carbide tools have higher abrasion resistance and hot hardness than the other tools but are

*Bulletin II-A contains some information on machining chromium-nickel stainless steel castings.

less shock resistant. They are suggested for continuous cuts using heavy equipment. The carbides are used on rigid automatic machines for light finishing cuts with box and balance turning tools, for forming, skiving and cutting off.

CUTTING FLUIDS

Petroleum base oil, water soluble and chemically active water base cutting fluids are used in machining stainless steels. Chlorinated, sulfurized fatty mineral or sperm oils are extensively used for automatic machining. Where thinning of the oil is desirable, it may be diluted with paraffin or light mineral oil. The heavy duty water soluble oils and chemically active fluids find greatest use on engine lathes, drilling and milling machines.

DRILLING

High speed steel drills are normally used for drilling austenitic nickel-containing stainless steel. Drills having a surface treatment to improve their abrasion resistance are desirable. Drills should be as short as possible to accent rigidity. The use of heavy web drills assists rigidity.

When possible, a drill jog bushing should be incorporated in the setup. The bushing should be short so as to permit the use of short drills. A distance of at least one drill diameter is necessary between the bushing and work to facilitate chip removal.

It is good practice to periodically back a drill out of the hole to clear chips and assist the flow of cutting fluid. A suggested sequence after an initial penetration of two or three drill diameters is to drill one or two diameters and thereafter back out after each penetration of three-quarters diameter. The drill should not dwell before backout or upon resumption of drilling.

Drill Geometry

Drills of standard helix ground to an included point angle of 130 to 140 degrees, a lip clearance of 6 to 9 degrees and a chisel point of 115 to 125 degrees are satisfactory for drilling normal depth holes of three times the drill diameter. However, where the steel is considerably below a hardness of 250 Brinell a drill point angle of 130 degrees, lip clearance of 12 degrees, and chisel point of 115 degrees is suggested. When possible, the web at the chisel point of the drill should be thinned to one-half the web thickness of a new drill. The thinned portion on the lip of the drill should be about one-third but not more than one-half the lip length.

A slight, narrow groove ground in the faces of the lips of the drill from and parallel to the cutting edges increases drill rake and aids in curling the chips. Decreasing the rake angle tends to break the chips into smaller pieces; however, the rake angle should not be negative.

Generally, the drilling of holes to a depth exceeding three times the drill diameter is considered a deep hole drilling operation. When the holes are drilled with a helical drill, a crankshaft type drill point grind with the offset notch point is suggested. The included point angle of the drill can be adjusted to the specific job. The overall grind suggested is an included angle of 135 degrees, lip clearance 9 degrees, notching angles of 55 degrees at the lip heels forming a chisel edge of about 120 degrees. The two notching cuts should just meet but not pass each other.

Drilling Small Diameter Holes

Speeds of 20 to 40 feet per minute are suggested for helical drills below 1/8 inch diameter. Operating such drills at higher speeds requires quite high revolutions per minute, which usually detracts from drill rigidity in addition to increasing machine vibration and tool breakage. The feed for drills below 1/8 inch diameter should be progressively less than .002 inch per revolution. A low feed that produces a chip, not powder, is optimum and the drill speed should be adjusted to the strength and load carrying capacity of the drill. It is important to back the drill out of the hole periodically to clear the chips and assist the flow of cutting fluid.

For drilling small diameter holes, commercially available heavy web drills of a somewhat higher helix angle than standard should be considered. For such service the drills are ground to an included point angle of about 135 degrees. They are also available with a notch-point grind that may perform best on medium and hard material.

Gun Drilling

Deep holes to 1 1/2 inch and somewhat larger in diameter are drilled with single flute gun drills. The choice of cutting tool material for nickel-containing stainless steel from an operational and economic standpoint is cemented carbide. Grade C-2 carbide is suggested. The remarks offered here are guidelines only, since manufacturers of tools and equipment for gun drilling have available much detailed information on this operation.

In addition to precision ground tools truly aligned with the work, gun drilling requires a precision starting bushing, a rigid equipment setup and an ample flow of clean filtered cutting fluid force fed under high pressure through the drill oil hole to the cutting tool.

Drill heads are available in solid carbide and as steel with a carbide insert. A suggested grind for the drill point is:

a) The outside diameter is ground cylindrical with .0006 inch per inch included back taper.

b) Outer lip width, 1/4 drill diameter.

c) Outer lip angle, 30 degrees.

d) Inner lip angle, 20 degrees.

e) Outer and inner lip primary relief on a $\frac{1}{32}$ inch land, 6 to 10 degrees.

f) Outer lip eccentric clearance below $\frac{1}{32}$ inch land, 12 to 15 degrees.

g) Inner lip straight clearance below $\frac{1}{32}$ inch land, 12 to 15 degrees.

Best chip control is obtained when the chips break apart at the apex of the drill point between the outer and inner lip. This usually occurs with medium hard and free machining materials. This may not occur in ductile material when the cutting angles are too high and the included apex angle is too small. Thus, to assist in chip control it is advisable to increase the included angle by decreasing the outer and inner angles.

Operating Speeds

A speed range of 30 to 50 feet per minute encompasses drilling all grades of non-free machining material with conventional helical high speed steel drills. A like range for drilling free machining material is 50 to 80 feet per minute. However, as previously presented, small diameter holes below $\frac{1}{8}$ inch diameter are usually drilled at a lower speed, namely 20 to 40 feet per minute, because of vibration and tool breakage at high revolutions per minute.

Tables I to IV present additional suggestions for helical drills and for gun drilling.

TABLE I

Permissible Feeds for Helical Drills

Diameter of Drill, in.	Drill Feed, ipr
Under $\frac{1}{8}$	.001–.002
$\frac{1}{8}$–$\frac{1}{4}$	.002–.004
$\frac{1}{4}$–$\frac{1}{2}$	.004–.007
$\frac{1}{2}$–1	.007–.012

TABLE II

Reductions in Speeds and Feeds for Helical Drills as the Depth of Hole Increases

Depth of Hole	Reduction of Speed, %	Reduction of Feed, %
3 times drill dia	10	10
4 times drill dia	20	10
5 times drill dia	30	20
6 to 8 times drill dia	35 to 40	20

TABLE III

Feeds for Helical Drills on Several Types of Machine Tools

Type of Machine	Drill Diameter, in.	Drill Feed, ipr
Heavy Duty Turret Lathe	$\frac{1}{2}$–1 Over 1	.003 –.005 .005 –.010
Heavy Duty Single and Multiple Spindle Automatics	$\frac{1}{2}$–1	.005 –.010
Single Spindle Automatic Screw Machines	$\frac{1}{16}$–$\frac{5}{8}$	.0015–.006

TABLE IV

Suggested Starting Speeds for Gun Drilling Material of 180 to 200 Brinell Hardness

Drill Dia, in.	Cutting Speed,[a] fpm	Drill Feed, ipr
$\frac{1}{8}$	170–200	.00021
$\frac{1}{4}$	"	.00039
$\frac{1}{2}$	"	.0006
$\frac{3}{4}$	"	.0008
1	"	.001
$1\frac{1}{2}$	"	.0015

[a] The lower speed of the range is for non-free machining material.

REAMING

Austenitic stainless steels are reamed with straight and helical fluted high speed steel and cemented carbide reamers. The latter usually produce the best finish.

Left hand helical fluted, right hand cut reamers are generally used for reaming straight holes. They are also used for an interrupted cut such as a keyway. Straight fluted reamers are suggested where extreme accuracy is required. A right hand helical fluted, right hand cut reamer cuts freely but if there is lost motion in the machine spindle, removal of too much stock or misalignment with the work, and a weak tool-work setup, it will pull itself into the hole and dig into the work.

The machine setup should include a full floating reamer holder. Reaming speeds and feeds are generally governed by the size of the reamer and the strength and hardness of the workpiece. The reamer feed should be sufficient to produce a small chip.

For finishing a tapered hole, taper turning with a single point tool is more satisfactory than taper reaming. If reaming is done, the hole should be prepared with a step drill or by step boring. Reaming is then done at a slow speed. A left hand spiral fluted (about 45 degrees), right hand cut reamer is recommended.

A stock allowance of .005 to .007 inch on diameter is suitable for finish reaming holes up to ½ inch in diameter while .012 to .015 inch allowance is recommended for larger holes.

Reamer Geometry

Primary relief angles are presented in the following suggested reamer geometry. Secondary clearance should be applied where necessary to eliminate tool drag. A short length secondary chamfer with clearance to a sharp edge immediately back of the first chamfer improves dimensional tolerance and finish.

Chamfer Angle	30°
Chamfer Relief	8°
Radial Rake	5°
Peripheral Clearance or Radial Relief	Decrease with increase in reamer diameter, i.e., 20° to 8° for ¼ to 1½ inch diameter.
Margin Width	Increase with increase in reamer diameter, i.e., .005 to .014 inch for ¼ to 1½ inch diameter.
Back Taper	Approx. .0002 inch per inch.

Operating Speeds

The suggested speed range encompassing all grades of material reamed with high speed steel reamers to close tolerance and a smooth finish is 15 to 40 feet per minute. In instances where sizing for dimensional tolerance takes precedence over finish, and rigidity without chatter of the reamer can be maintained, a speed range of 40 to 80 feet per minute is satisfactory for non-free machining material and 40 to 120 feet per minute for free machining material.

Grade C-2 cemented carbide tipped reamers perform well. Carbide reamers are used principally because they produce a good finish and afford long tool life. Generally the best results are attained with carbide reamers when they are not operated at excessive speeds. Suggested speeds may be similar to or moderately higher than those used with high speed steel reamers.

Table V presents guidelines on feeds for high speed steel and cemented carbide reamers. They are considered as feeds for finish reaming but should be adjusted to meet the requirements of specific jobs.

TABLE V

Finish Reaming Feeds

Reamer Diameter, in.	Number of Reamer Flutes	Feed per Revolution, in.	Feed per Reamer Flute per Revolution, in.
⅛ and under	4	.002–.003	.0005–.00075
¼	6	.005	.00083
½	6	.007	.0012
1	8	.012	.0015
1½	10	.016	.0016

TAPPING

Precision ground high speed steel taps are used for tapping nickel-containing stainless steel. Those having a surface treatment to improve wear resistance are preferred.

Taps below ½ inch diameter or, in some instances, ¼ inch diameter are produced with concentric lands except on a few incomplete threads of the chamfer. These taps without radial land relief do not perform as freely on nickel-containing stainless steel as those with relief. To assist free cutting, the lands should be narrowed and some relief ground at the heels of the lands when possible. When permissible, threads should not be produced beyond 75 per cent of full depth, particularly when the hole depth is more than 1½ times the diameter. This tends to lessen tap breakage. In many instances threads are tapped 50 to 60 per cent of full depth.

During tapping, metal may flow into the tap minor thread root diameter with resultant binding and thread tearing. This is most prevalent with non-free machining material in the soft condition. It can be avoided when preparing the holes for tapping by the use of sufficiently large drills and reamers.

Smooth reamed holes are preferred for tapping. Best quality threads are produced when the tap is mounted in a rigid, properly aligned holder. Where accurate alignment cannot be maintained, the tap should be mounted in a full floating holder. Tapped threads are of better quality when the tap is lead screw controlled.

Low quality threads and tap breakage may result from the use of machines which are too light or too heavy relative to the tap diameter. For instance, a machine that handles a ¾ inch tap is usually found to be too heavy and insensitive for a No. 10 tap.

Two-fluted taps are suggested to tap size number 6, three-fluted from size 6 to ½ inch diameter and four-fluted above ½ inch.

Regular taper, plug and bottoming hand taps are used for hand tapping unless the job presents unusual conditions. For instance, serial hand taps are suggested for tapping deep holes and material of high hardness.

Spiral fluted taps are helpful for ejecting chips from the hole entrance; also for tapping a hole with an interruption such as a keyway. For cutting a right hand thread the tap should have a right hand spiral.

Short-flute spiral pointed taps are generally preferred for automatic production.

Interrupted thread taps of plug and spiral point design are suggested for deep holes. They may be helpful when trouble arises on general machine tapping.

Tap Geometry

Tap chamfer lengths in terms of threads from the root diameter of the end thread on taper, plug and bottoming taps are 8 to 10, 3 to 5 and 1 to 1½, respectively. Approximate mean corresponding chamfer angles in degrees for American National and Unified thread forms are 4½, 11 and 31 degrees.

When bottom tapping to a thread height of, say, 60 per cent, a 1½ thread should be used if possible. This permits the largest allowable chamfer end diameter, a lower chamfer angle and more tap cutting teeth.

Eccentric relief taps with a 15 degree face hook angle and 12 degree radial chamfer relief perform satisfactorily.

The spiral point angle of a spiral pointed tap should be ground to the angle supplied by the tap manufacturer. This is usually 15 to 20 degrees.

Narrowing the width of thread land on the tap by grinding down its heel usually affords freer cutting and lessens metal pickup.

Freehand grinding of taps should be avoided if at all possible. The accuracy of grinding the chamfer and its relief is of utmost importance since it affects the quality of the tapped thread. Tap chamfers and their relief should be ground on a precision tap chamfer grinder. When set up for grinding, the tap should rotate in true concentricity.

Operating Speeds

The overall speed range for tapping the various stainless grades is 10 to 25 feet per minute using the higher speeds for the free machining alloys.

Pipe Threads

Tapping pipe threads is similar to tapping conventional screw threads. Straight and tapered pipe threads usually have sharper roots and crests and are tapped to a greater depth than screw threads. This imposes a restraint, especially with taper threads. Here practically every tooth is cutting, whereas on the straight pipe thread nearly all cutting is done by the chamfered teeth. Interrupted thread taps are suggested if difficulties are experienced with soft material. Normally there is less metal pickup with interrupted taps and, consequently, less wear. Taper reaming prior to tapping tapered holes is preferred. Generally the lands of taper taps are radially relieved to the tooth face. Receding chaser collapsible taps are suggested for tapping taper threads when they are available in the size of hole to be tapped. An independent positive feed is recommended with this type tap because the receding action of the chasers largely removes their self-leading action.

THREADING

Production threading of straight threads is accomplished with tangent, circular, milled and tapped high speed steel thread chaser inserts mounted in self-opening die heads. However, a solid adjustable die head with removable chaser inserts is suggested for small diameter machine screws below size 10. Acorn and split or button dies are not generally recommended for machine threading. Receding chaser die heads are recommended for taper threading when the diameter to be threaded is within the sizes in which the heads are produced. With all head types, best quality threads are obtained with a positive feed.

Chaser Insert Grinds

The following remarks on thread chaser grinds have proven applicable to threading nickel-containing stainless steel. Nevertheless, each job has its peculiarities and one should consult with the manufacturers of threading equipment or refer to their literature for details. The rake angles cited may be decreased somewhat for free machining material.

The chaser's chamfered threads or throat angle forms the thread and is required to remove excessive material from oversize stock. Oversize stock should be avoided because it imposes high cutting loads on the chamfered teeth. Metal flowing into the roots of the chaser threads due to oversize stock or the plastic flow of soft material can cause similar difficulty. In either instance, reducing the stock diameter before threading is helpful. Chamfering the end of the stock aids the chasers to start cutting.

A chaser throat angle with at least three chamfered threads is preferred. Usually a throat angle with two chamfered threads is satisfactory but the chip per tooth with a longer chamfer and more teeth is thinner, affording lower cutting forces and better quality threads. A throat angle with 1½ chamfered threads is suggested for close-to-shoulder threading. The approximate throat angles for 1½, 2 and 3 chamfered threads for American National and Unified thread forms are respectively 33, 22 and 15 degrees. For Acme and similar thread forms a throat angle of about 12 degrees is suggested. Generally the throat angle of tangent type chasers need not be reground because they are supplied to standard lengths.

Tangent chasers are ground to a rake angle of 25 degrees. The lead angle of the chaser is the angle made by the end of the chaser with the center line of the piece. The lead angle varies with the helix angle of the thread, with the type of die head and with or without the use of a lead-screw feed for the die head. Lead angles should be obtained from literature of the die head manufacturer.

Generally, tangent chasers produce threads to the closest tolerance. They are quite adaptable to cutting form threads such as Acme and pipe threads. Receding die heads are preferred over jam threading (cutting across the full width of the chaser) for the production of tapered threads.

Circular chasers are ground to a rake or a chordal rake angle of 20 to 25 degrees, a face angle of 1½ degrees and a chip clearance angle of 12 degrees.

Milled and tapped insert chasers are ground to a rake or chordal rake angle of 10 to 15 degrees. Increasing the clearance on milled inserts at the chamfer or throat from the chaser face to the heel, and on tapped inserts at the heel over the chaser width, usually affords freer cutting.

Operating Speeds

The speed range for general threading operations on Unified and American straight threads is 10 to 25 feet per minute; the maximum speed is applied to free machining material, while the lower speed is used for non-free machining material. Usually, equivalent quality fine series threads can be produced at a somewhat higher speed than coarse threads.

Speeds of 8 to 12 feet per minute are used for producing taper and form threads.

Where high quality threads are desired, a speed of 10 feet per minute (and as low as 5) is suggested.

Stainless steel is adaptable to thread rolling, and high speed steel dies are suggested. Tangent rollers are preferred over bump rolling.

Conventional cylindrical and flat reciprocating dies are used for secondary threading operations.

Each end of the part to be threaded should be chamfered to 30 degrees. This will aid the rollers to start and minimize their chipping at start and finish.

Applications of thread rolled parts should be avoided where work hardening from thread rolling could lead to stress corrosion cracking in service.

MILLING

High speed and cobalt high speed steels are ordinarily used for milling these alloys. Cobalt high speed steel is preferred because of its higher wear resistance. The following remarks apply to steel cutters; however, reference is made to the use of cemented carbide face mills.

Climb (down) milling is suggested where possible rather than conventional (up) milling. Climb milling requires sturdy machines and positive feed without backlash.

Cutters

Coarse tooth cutters of 25 to 45 degree helical teeth are preferred for plain and slab milling. These are termed heavy duty cutters. If not available in the width to be milled, light duty cutters with straight teeth are suggested.

Staggered tooth cutters with alternate teeth of opposite helix are efficient for milling slots in widths from 1 to $\frac{3}{16}$ inch. Interlocking cutters of like tooth design are used for wider widths.

Metal slitting saws with sides relieved or dished are used for cutting off and milling slots in widths below $\frac{3}{16}$ inch.

Narrow width cutters and saws should be supported by mounting them on the arbor between flanges provided with a driving means other than the standard key. The flanges should support the cutters to the maximum permissible diameter.

End mills with 30 to 35 degree helical teeth are preferred because they cut with a smoother shearing action. Two-lip end mills with end teeth cut to the center are suggested for slotting. For milling slots where the end of the cutter is in contact with the work, it is usually preferred to use a cutter of the same hand of cut and hand of helix. When profiling with the periphery of a cutter and when the end teeth are not in contact with the work, a cutter of right hand cut and left hand helix or left hand cut and right hand helix should be used.

Shell end mills of 18 to 30 degree helical teeth are used for face milling surfaces at right angles to each other.

Heavy duty rather than light duty face mills are suggested as they are more rugged with fewer teeth. Mills with high speed steel or cemented carbide inserts are satisfactory.

Cutter Grind

The included tooth angle between the face of a cutter and the land should be as large as possible to give support to the cutting edge. Additional support to the cutting edge is obtained by providing the land with clearance as low as possible but sufficient to cut freely. A useful guide for providing suitable primary clearance and land width is to apply to ⅛ to 3 inch diameter cutters with 13 to 5 degree clearance, respectively, decreasing proportionately as the diameter increases. Apply 4 to 5 degrees to cutters over 3 inch diameter. Use a land width of 1/64, 1/32 and 1/16 inch on small, medium and large diameter cutters, respectively. Secondary clearance of 3 to 5 degrees larger than primary clearance is satisfactory.

Provide side teeth of cutters, such as side mills, and slitting saws with radial relief to prevent binding between the teeth and work. The side teeth of saws may be dished to provide relief. Generally a relief angle of 3 to 5 degrees is satisfactory for milling cutters and 1 to 2 degrees for saws.

When face milling with a shell end or a conventional face mill, it is best to have a chamfer of sufficient width that confines the cutting along the chamfer. If not, undesirable chip flow might exist when the cutting edge of a narrow chamfer and the peripheral cutting edge of the cutter are simultaneously cutting. A square cutter nose should be avoided since it usually breaks down quickly. When milling to a square shoulder endeavor to use a small chamfer or radius of about 1/16 inch.

Excessive rake is dangerous when milling thin sections because of the tendency of the cutter to bite into the work.

Plain profile-relieved cutters with teeth on the circumferential surface and in various widths for milling flat surfaces, and shell end mills, are ground to a radial rake of 10 to 15 degrees. Plain form ground cutters are provided with a radial rake of 5 to 10 degrees. Slitting saws with 5 to 10 degrees radial rake are satisfactory.

Suggested geometries for end and face mills are given in Tables VI and VII.

Operating Speeds

Economic tool life is attained by adjusting the peripheral cutting speed of the cutter according to the tensile strength and hardness of the workpiece. Although material of low mechanical properties machines with a thicker, rougher chip, it can usually be machined at speeds above those used in cutting higher strength material. A low cutting speed is always desirable for long tool life. A milled finish usually improves with higher cutting speed, but longer cutter life is attained by lowering the feed. The combined effect of increasing the speed and lowering the feed is a compromise to attain a good finish.

A depth of cut of 1/8 inch or more is considered a roughing operation, while a cut several thousandths to 1/16 inch deep is considered a finishing operation.

Avoid excessive as well as insufficient feed. The former physically and thermally overloads the cutter teeth while the latter permits the cutter to glaze and work-harden the material.

Due to the many factors affecting cutter tooth load (such as tool design and sharpness, rigidity of machine, work material hardness, etc.), it is not practical to offer more than guidelines on the feed per tooth. For example, with a narrow saw the feed might be 1/8 that suitable for other type cutters. Generally the feed should be lowered with increase in depth of cut and decrease in width and saw diameter. Table VIII contains guidelines on cutter feeds for non-free machining stainless steel.

Peripheral operating speeds for high speed steel plain, end and face milling cutters on free machining

TABLE VI

Primary Clearance Angles for High Speed Steel End Mills for Free and Non-Free Machining Materials

Helix angle	30 to 35°	End clearance angle	3 to 7°
Radial rake	15°	Peripheral clearance angle	10°
End cutting edge angle	3°	Corner angle	45° x 1/16 in.

TABLE VII

Primary Relief Angles for High Speed Steel and Cemented Carbide Face Mills

Work Material	Free Machining		Non-Free Machining		Free and Non-Free Machining	
Tool Material	HSS	CC	HSS	CC	HSS	CC
Primary Relief Angles						
Axial rake, deg	0	5 to 11	0 to 5	5 to 11	3	7
Radial rake, deg	0	-5 to -11	0 to 5	-5 to -11	3	-7
Corner angle, deg	45	45	45	45	45	45
Face cutting edge angle, deg	5	5	5	5	3	3
Face relief angle, deg	8 to 10	8 to 10	8 to 10	8 to 10	5 to 7	4 to 5
Corner or Peripheral relief angle, deg	8 to 10	8 to 10	8 to 10	8 to 10	6 to 10	3 to 4

TABLE VIII

Cutter Feed for Non-Free Machining Material[a]

Tool Material	Cutter Feed, in./tooth, for Cutter Type Shown					
	Plain	Slotting and Side	Circular Saw	End Mill	Form Relieved	Face Mill
HSS	.004	.005	.001	.003	.002	.006
CC	—	—	—	—	—	.005

[a] Feeds may be increased 30 to 40 per cent for free machining material.

material range from 60 to 115 feet per minute. The speed for non-free machining material ranges from 40 to 85 feet per minute. Narrow saws and saws above about six inches in diameter perform best at speeds towards the lower end of the speed range. Like speeds are suggested for deep slotting and side milling. The suggested speed range for cemented carbide insert tooth face mills is 250 to 300 feet per minute. In each foregoing instance the minimum of the speed range is intended for material above 270 Brinell hardness.

TURNING, BORING AND CUTTING OFF ON ENGINE AND TURRET LATHES

Turning

Engine and turret lathes are usually considered for single point tool operations.

The geometry of single point turning tools is presented in Table IX. Optimum chip control is afforded by a chip breaker. Mechanical chip breakers are used with carbide inserts. The breakers are machine ground into the high speed steel and cast nonferrous tools.

Table X suggests overall speeds for turning operations.

TABLE IX

Single Point Turning Tool Geometry

Work Material	Tool Angles, deg					Nose Radius, in.
	Back Rake	Side Rake	End and Side Relief	End Cutting Edge	Side Cutting Edge	
	High Speed Steel and Cast Nonferrous Tools					
Free Machining	5	8	6	5	a	b
Non-Free Machining	0	15	6	5	a	b
	Cemented Carbide Tools					
Free and Non-Free Machining	0 –5[c]	5 –5[c]	5 5	5 5	a a	b b

[a] When possible use a side cutting edge angle of at least 15 degrees.

[b] Use a nose radius of 1/32 inch for cuts 1/8 inch or less in depth. For cuts 3/16 to 3/8 inch inclusive use a radius of 3/64 inch.

[c] Geometries suitable for either type material; however, the negative rake should be used for interrupted cuts.

TABLE X

Cutting Speeds for Turning with Single Point Tools

Work Material	Cutting Tool Material	Type of Cut and Speed, sfpm	
		Roughing[a]	Finishing[b]
Free Machining	HSS	70–90	100–140
	Cast Alloys	100–150	150–200
	Cemented Carbide	150–250	200–400
Non-Free Machining	HSS	60–90	100–120
	Cast Alloys	100–130	100–150
	Cemented Carbide	130–180	150–300

[a] For cuts to 1/4 inch deep at .010 inch minimum feed per revolution. The minimum speed of each range is for maximum depth cuts.

[b] For light cuts at feeds below .010 inch per revolution. The maximum speed of each range is for minimum depth cuts.

Boring

Tool materials used for boring are of the same type as those used for turning. Boring bars should be as short as possible or equipped with a pilot support affixed to the machine head stock to afford rigidity. The depth of cut for boring should be lower than used for external turning. The speeds listed in Table X can be used as a guide for boring. It is suggested that boring operations start at the minimum surface speeds noted in Table X with a feed commensurate with the desired finish. It is well to consider that an increase in depth of cut or feed necessitates a decrease in cutting speed but the higher speed affords a cleaner cut surface with less buildup on the tool.

Cutting Off

Cutoff tools are of the same materials as those used for turning and boring. The suggested minimum tool width is $\frac{1}{16}$, 0.100, $\frac{1}{8}$ and 0.140 inch for work diameters of $\frac{1}{4}$, $\frac{9}{16}$, $\frac{3}{4}$ and 1 inch, respectively. Where possible, these widths should be increased to attain rigidity. The cutting edge of straight or circular cutoff tools may be straight or provided with an end cutting edge angle of 5 to 15 degrees. An angle of 10 to 15 degrees is suggested for work diameters to about $\frac{3}{4}$ inch. Proportionately smaller angles for 5 degrees are used for larger work diameters. This angle is not commonly used on engine and turret lathe tools but is recommended for machining on automatic machines.

The straight cutoff blades are ground to 2 to 3 degrees side relief. This relief should be provided on circular tools. The suggested end relief is 7 to 9 degrees. If top rake is desired a shallow saucer form is ground into the top face of straight tools and an angle equivalent to it in circular tools.

Moderate surface speed is suggested approaching the minima of the speed ranges cited in Table X. The feed should also be moderate, for instance .0005 to .0015 inch per revolution for $\frac{1}{4}$ inch diameter work material, increasing to .003 inch for 1 inch diameter stock.

AUTOMATIC MACHINING WITH GENERAL PURPOSE HIGH SPEED STEEL

Nickel-containing austenitic stainless steels are extensively machined on automatic machines. The previous remarks on the geometry of cutting tools apply to tools for automatics.

Box Tools

Box tools are ground to cut with a shearing action. Chip curlers or breakers are ground into the top face of tools. V-groove type curlers are generally ground in high speed steel and cast alloy tools. Parallel breakers are ground into cemented carbide tools. The tools are ground to afford a back rake (when mounted in their holder) of 8 degrees, side rake 10 to 12 degrees and end clearance 3 to 5 degrees.

When box tool turning these alloys, roller rests are preferred to V-rests to support the work as they are less prone to galling and severely work-hardening the work. The rollers of the rest are set slightly behind the cutting point of the tool.

Box tools on single spindle automatic screw machines are operated on free and non-free machining materials at 90 to 125 and 60 to 90 surface feet per minute, respectively. Feeds range between .002 to .005 inch per revolution. The lower feeds are used for finishing and heavy cuts. The suggested cutting speeds on heavy duty single and multiple spindle machines are 90 to 115 and 60 to 80 feet per minute, respectively. The feed ranges from .005 to .010 for roughing and .003 to .005 inch for finishing.

Form Tools

Axial clearance is not usually applied to circular form tools as small chips may enter and load the clearance space. To compensate for the lack of clearance a slight taper of 1 to 3 degrees can be applied to shoulders of the tool. The excess stock due to the taper can be removed on a secondary operation with a shaving tool.

The width of a form tool, especially a roughing tool, should not exceed $1\frac{1}{2}$ times the stock diameter. If a wider form must be made, consider a flat roughing tool with normal side relief made in two sections and separately mounted. The cutting speed and feed of a form tool should be decreased with increase in its width.

Unless the form of the tool is designed for top rake, the exact form of the tool will not be machined into the work. However, little change results from a low rake angle of 4 to 8 degrees.

When possible, all sharp corners should be removed from roughing form tools.

Form tools are operated within the range of surface speeds used for box tools. A feed up to .001 inch per revolution is suggested for roughing on single spindle machines and to .0004 inch for finishing. The feed on heavy duty single and multiple spindle machines ranges to .003 inch for roughing and to .001 inch for finishing.

FRICTION SAWING

Friction sawing is performed on a heavy duty vertical type band saw. The saw band operates at a very high velocity and sawing is accomplished by the friction and heat resulting from the saw. The developed heat softens the material and permits the saw tooth to cut away a soft chip.

Flexible back, hard tooth saw bands of alloy steel with raker set teeth of conventional zero rake are used for straight and contour sawing. Saws with 10 to 14 pitch teeth are suggested. Fourteen pitch saws are used only for thin work.

The maximum thickness of material suggested for production sawing is ¾ inch. Larger work sections have been sawed by the rocking feed technique. Nevertheless, it is well to determine from producers of friction sawing machines the sizes suitable for thicknesses above ¾ inch.

Sawing stacked work should be avoided. Chip droplets at the kerf of the cut from between the stacked parts may cause difficulty by fusion welding or otherwise.

To successfully saw tubing the relationship between its outside diameter and wall thickness must be within certain limits. These data are presented in the literature of saw manufacturers. Briefly, the aforementioned relationship should be such that as the saw breaks through the tube inner wall the thickness of cut (with the vertical axis of tube) should not exceed 1 inch.

Twisting and breaking of the saw band can result from improper setting of the saw guides or from too much tension in the band. Many machines are equipped with band tension indicators calibrated to band width. In any event, proper tension is an important consideration.

For saw band rigidity and accuracy of cut the widest and thickest band possible should be used. Where a small radius is to be cut, especially in thick material, it is best to make the cut in more than one operation, using a narrower width and possibly a lower gage band for each cut.

Light to moderate feed pressure should be employed remembering that higher saw velocities and feed pressures are required with increase in work thickness. Saw band velocities range from 7000 to 14,000 feet per minute. In the absence of in-plant experience on operating speeds, one should refer to literature of the producers of saws.

BAND SAWING

Horizontal machines are used for cutting off bar, plate, structural shapes, pipe, etc. They are also adaptable to sawing stacked work material. The saw band is fed into the work by automatic feed.

Vertical machines are generally used for splitting, slotting, removing excess stock from parts, contouring and thin wall tubing. Some vertical machines are equipped with automatic feed; however, in the absence of automatic feed the work may be manually fed into the saw.

Flexible back, hard tooth saw bands of high speed steel or alloy steel are used on both type machines. High speed steel bands are normally used on horizontal machines. Alloy steel bands are usually operated at the same speeds and feeds as high speed steel bands but they afford lower tool life.

The number of teeth per inch of the saw (the pitch) is governed by the thickness of the work material. Fine pitch saws are used for sawing thin sections and the coarseness of the pitch increases with section thickness. For satisfactory cutting there should always be at least two teeth in contact with the work. As a guide on the selection of the pitch of saws, 32, 14 and 10 pitch teeth, respectively, might be used for 1/16, ¼ and ½ inch thick work sections.

Raker set teeth are usually satisfactory for cutting off on horizontal machines. They are usually satisfactory on vertical machines; however, if they offer difficulty when sawing thin sections and sections of varying cross section, wave set teeth are suggested.

Both regular and hook tooth forms are used as indicated in Tables XI and XII.

The remarks under Friction Sawing on saw band tension, width, thickness and radius cutting apply equally to band sawing.

The feed force for a fine pitch saw and thin work material should be lower than for a coarse pitch saw and heavier work material. The feed force at a specific band velocity governs the cutting rate (square or linear inches per minute). For hollow material such as tubing, the rate should be 40 to 70 per cent of that used for solids of the same cross sectional area. The maximum reduction is applied to sections ⅜ inch and under and the minimum to sections ⅝ inch and above. Generally, the feed force should be decreased with decrease in work thickness and the band velocity increased with decrease in work thickness. Lower feed force and band velocity is usually required for higher hardness work materials.

Saw band velocity and feed force for each job must be cautiously adjusted to attain economic production

TABLE XI

Guidelines on Cutting Speeds and Cutting Rates for Cutoff Sawing on Horizontal Machines with High Speed Steel Saw Bands[a]

Type of Stainless Steel	Brinell Hardness	Saw Band Velocity, fpm	Cutting Rate, sq in. per min
303, 303Se	150–200	90-130	5–2
201, 202, 302, 304	130–190	80–120	4–2
308, 309, 310	160–220	60–80	2–1
314, 316, 317	160–220	50–75	2–1
321, 347	165–200	90–120	4–2

[a] Based on the use of 1 inch wide high speed steel band, regular tooth form (except hook tooth form for metal thicker than about 10 in.), raker set teeth, to cut solid bar stock up to 18 in. thick.

TABLE XII

Guidelines on Cutting Speeds for Band Sawing on Vertical Contour Machines with High Speed Steel Saw Bands

Type of Stainless Steel	Brinell Hardness	Saw Band Velocity, fpm, for Various Stock Thicknesses		
		1/4 to 1/2 in.[a]	1 to 3 in.[b]	6 to 12 in.[c]
303, 303Se	150-200	160	130	100
201, 202, 302, 304	130-190	150	100	70
308,* 309,* 310*	160-220	110	80	50
314,* 316,* 317*	160-220	95	60	40
321, 347	160-200	150	100	70

[a] Regular tooth form; 10 pitch, minimum feed force, except average for steels marked with an asterisk.
[b] Regular tooth form; 6 pitch, average feed force, except maximum force for steels marked with an asterisk.
[c] Hook tooth form; 3 pitch, maximum feed force.

and to avoid premature saw dulling and/or breakage. Although it is understood that narrow width and low gage saws will not withstand the cutting pressure of heavier saws, this point must always be considered. In the absence of data on saw band velocities and cutting rates, guidelines are given in Tables XI, XII and XIII.

TABLE XIII

Guidelines on Linear Cutting Rates with High Speed Steel Saw Bands on Vertical Contour Machines

Work Thickness, in.	Cutting Rate, linear in. per min
1/4	7 3/16
1/2	4 3/4
1	2 5/8
1 1/2	1 5/8
3	5/8
6	5/16

[a] For Types 308, 309, 310, 314, 316 and 317 decrease cutting rates 40 per cent when setting up a job to arrive at the optimum rate.

When setting up a job refer to Table XI or XII for the saw band velocity range. Choose a speed based on the hardness of the work materials, thickness, type of cross section and the geometry of the saw band. Operate at a low to moderate cutting rate and time a length of cut. If working on a horizontal machine, compute the square inches of material cut per minute or, if working on a vertical machine, determine the linear inches cut per minute. Providing the hardness of the work material is within the range noted in Tables XI and XII, increase the cutting rate to within the range cited in Table XI or XIII. In any event, adjust the feed to cut a clean, curled chip.

HAND AND POWER HACKSAWING

High speed steel or flexible back blades with high speed steel teeth are used for hand and power hacksawing. Saw blades of 14 to 18 raker set teeth per inch are suitable for hand sawing. However, tubing of wall thickness below 1/16 inch is sawed best with saws of 18 or more wave set teeth according to the thinness of the tube wall.

Whether the operation is by hand or power, the blade should never be allowed to ride the work. Maintain positive feed and raise the blade from the work on the return stroke.

Blades of 10 to 14 raker set teeth per inch are suggested for power sawing stock to 3/4 inch. Coarser pitch blades of 6 to 4 raker set teeth per inch are used for material above 3/4 inch thick. Where difficulty is encountered on sawing work below 1/4 inch thick, or tubing, a blade with wave set teeth of 14 pitch (18 tpi if available) might be useful.

Wide blades of heavy gage are best for power hacksawing. Short, heavy blades cut accurately, afford the best tool life and are less prone to break. Blades should be mounted with proper tension. When tension is determined by the ring and feel method, the blades should be tensioned until they produce a dull thud when struck with a tool.

The speeds for power hacksawing austenitic nickel-containing stainless steels are principally influenced by the hardness of the work. As a guide, annealed material to about 185 Brinell hardness is sawed at 100 strokes per minute, and cold drawn material to 275 Brinell hardness at 80 strokes. Moderate feed pressure is used for soft material, light cross sections and increased number of teeth per inch of saw. Increased feed pressure is used on harder material, heavier sections and a decrease in the number of teeth per inch of saw.