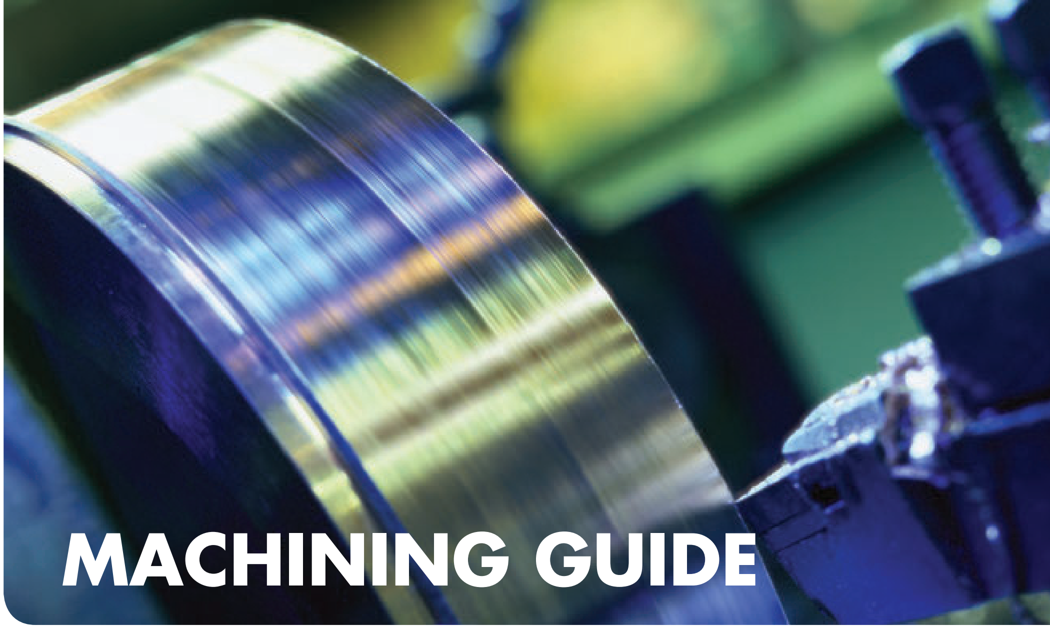
ED FAGAN INC.
Special Purpose Alloys



MACHINING GUIDE

Nickel-Iron Alloys

Molybdenum & Molybdenum Alloy

Tungsten Alloys

Soft Magnetic Alloys

THE MATERIALS YOU NEED, WHEN YOU NEED THEM

ED FAGAN INC.

General Guide to Machining Nickel-Iron Alloys

The Nickel-Iron Alloy Family generally includes INVAR, KOVAR®, Alloys 42, 46, 48, and 52, Alloy 42-6 and the magnetic shielding alloys such as MuMetal.

CHARACTERISTICS

This group of alloys is not hardenable by heat treatment. They can be made harder through cold working only. The annealed hardness for these alloys is generally in the range of RB 70/80, whereas the 1/4 H to 1/2 H range for this group of metals, can run between RB 80/96. Material in the annealed condition will be more difficult to machine because it is soft and gummy. The tools tend to plow the metal instead of cutting into it, and do not easily form chips. Surface scale oxide is tightly adherent and penetrates the surface to a greater extent than stainless steels. Machining is considerably improved by descaling the material. If there were standard machinability ratings applied for this series of alloys, Alloy AISI-B-1112 being measured as 100%, the following percentages could be suggested for these chemistries:

INVAR 36 FM (ASTM F-1684) – 60%
KOVAR (ASTM-F15) – 40%
ALLOY 48 (ASTM F-30) – 40%

COOLANT

It is important to control heat buildup, the major cause of warpage. Suggested coolants are Keycool 2000 or Prime Cut. Whatever lubricant is used for machining, it should not contain sulphur. Sulphur can effect the performance of many sealed electronic parts.

TOOLING

T-15 Alloy, such as Vasco Supreme - manufactured by Vanadium Alloys Company, M-3 Type 2 such as Van Cut Type 2 - manufactured by Vanadium Alloys Company.

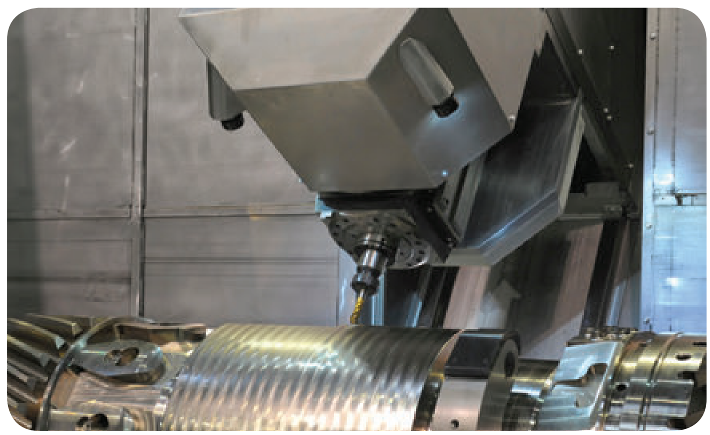
For machining with carbide tools, a K-6 manufactured by Kennemetal, Firthite HA manufactured by Firth Sterling, or #370 Carboloy could be used, or a K2S manufactured by Kennemetal, or a Firthite T-04 manufactured by Firth Sterling would be satisfactory. One thing of prime importance is that all feathered or wire edges should be removed from the tools. They should be kept in excellent condition by repeated inspection.

TURNING

If steel cutting tools are used, try a feed of approximately .010″ to .012″ per revolution and a speed as high as 35/ FPM could probably be attained. Some of the angles on the cutting tools would be as follows:

- End cutting edge angle - Approximately 7°
- Nose Radius - Approximately .005″
- Side cutting edge angle - Approximately 15°
- Back rake - Approximately 8°
- Side rake - Approximately 8°

When cutting off, high speed tools are better than carbide tools, and a feed of approximately .001″ per revolution should be used. The cutting tools should have a front clearance of about 7° and a fairly big tip - larger than 25° would be helpful.



DRILLING

When drilling a 3/16″ diameter hole, a speed of about 40/ FPM could possibly be used, and the feed should be about .002″ to a .0025″ per revolution, for a 1/2″ hole, approximately the same speed could be used with a feed of about .0040″ to .005″ per revolution. The drills should be as short as possible, and it is desirable to make a thin web at the point by conventional methods. By conventional methods, we mean do not notch or make a crank shaft grind. It is suggested that heavy web type drills with nitrided or electrolyzed surfaces be used. The hole, of course, should be cleaned frequently in order to remove the chips, which will gall, and also for cooling. The drill should be ground to an included point angle of 118° to 120°.

REAMING

Reaming speeds should be half the drill speed, but the feed should be about three times the drill speed. It is suggested that the margin on the land should be about .005″ to .010″, and that the chamfer should be .005″ to .010″ and the chamfer angle about 30°. The tools should be as short as possible, and have a slight face rake of about 5° to 8°.

TAPPING

In tapping, a tap drill slightly larger than the standard drill recommended for conventional threads should be used, because the metal will probably flow into the cut. It is suggested that on automatic machines, a two or three fluted tapping tool should be used. For taps below 3/16″, the two fluted would be best. Grind the face hook angle to 8° to 10°, and the tap should have a .003″ to .005″ chamfered edge. If possible, if binding occurs in the hole in tapping, the width of the land may be too great, and it is suggested that the width of the heel be ground down. Again, it is suggested that nitrided or electolyzed tools be used. Speed should be about 20/FPM.

HIGH SPEED TOOLS*

TURNING AND FORMING

CUT-OFF TOOL	SFM	FEED
1/16″	65	.0010
1/8″	67	.0012
1/4″	69	.0016
FORM TOOL	**SFM**	**FEED**
1/2″	67	.0012
1″	63	.0010
1 1/2″	63	.0009

DRILLING

DRILL DIA	SFM	FEED
3/8″	43	.0030
3/4″	45	.0036

MILLING

SFM	FEED
35-70	.002-.005

REAMING

SIZE	SFM	FEED
Under 1/2″	57	.0030
Over 1/2″	57	.0045

THREADING

T.P.I.	SFM
3-7 1/2	8
8-15	10
Over 16	16

TAPPING

T.P.I.	SFM
3-7 1/2	6
8-15	7
16-24	11
Over 25	16

BROACHING

SFM	FEED
8-12	.001-.005

TURNING SINGLE POINT & BOX TOOLS

TOOLS	SFM	FEED
High Speed	60-65	.0029
Carbide	160-215	.025-.080

**When using carbide tools, surface speed feet/minute (sfm) can be increased between 2 and 3 times over the high speed suggestions. Feeds can be increased between 50 and 100%.NOTE: Figures used for all metal removal operations covered are average. On certain work, the nature of the part may require adjustment of speeds and feeds. Each job has to be developed for best production results with optimum tool life. Speeds or feeds should be increased or decreased in small steps.The information and data presented herein are typical or average values are not a guarantee of maximum or minimum values. Applications specifically suggested for material described herein are made solely for the purpose of illustration to enable the reader to make his own evaluation and are not intended as warranties, either express or implied, or fitness for these or other purposes.*

GENERAL

Molybdenum and molybdenum alloys can be machined by all of the common metal machining processes. No special equipment or procedures are required to produce parts with accurate dimensions with excellent finishes. Molybdenum may be machined to achieve simple parts, very complex parts/shapes, and very small intricate parts.

TOOLS

The choice between high-speed steel and sintered carbide (C2 Grade) depends largely on production quantities. In either case, tool life is shorter than would be expected with steel because molybdenum is considerably more abrasive than steel at the same hardness, and molybdenum has a tendency to chip while being machined. High-speed steels are generally used for small quantities or for roughing cuts on uneven surfaces. The preferred carbide grades are the C2 types recommended for cast iron. Work should be firmly chucked; tools sharp and well supported; machines should be rigid, sufficiently powerful, and free from backlash.

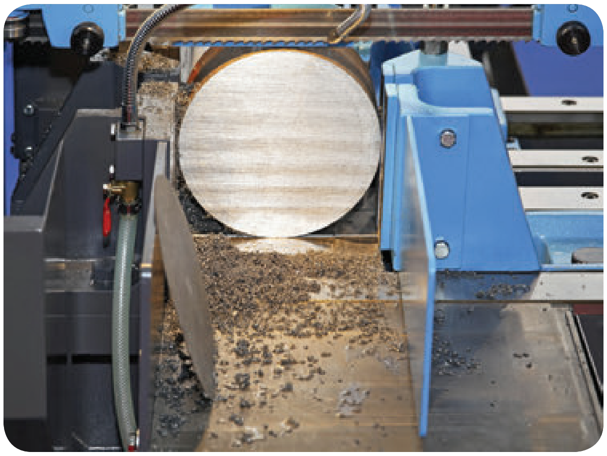
LUBRICANTS

Many types of machining are done without lubrication; but cutting fluids may be used to extend tool life, increase cutting speeds, remove heat from the tool and work-piece, and remove fine molybdenum particles that wear the cutting edges of tools. When a lubricant is used, various high-chlorinated oils and solvents have proved satisfactory.

Soluble oils are very effective in hacksaw and band saw cutting operations, but not effective for turning, drilling, reaming, or tapping. Sulfur-base oils and highly chlorinated cutting oils are very effective in drilling, tapping, or thread chasing; and some machinist use these oils for finishing cuts in lathe operations to yield a smooth bright surface. Highly chlorinated cutting oils are most effective in reaming operations. Sulfur-base cutting oils cannot be used in machining electronic parts because of their deleterious effect on final properties.

SAWING & SHEARING

Molybdenum saws readily with high-speed-steel band or hacksaws; the practice is similar to that normally used in superalloys and no coolant is necessary but use of a soluble oil coolant in the hacksaw or band saw cut will remove chips and lengthen blade life. High-speed steel blades with only the tooth area hardened are the most effective. Hand hacksawing is suitable for light gages only. On power hacksaws, cutting rates are about 80 strokes/min at 0.004-in feed; and on band saw equipment, 100 fpm with a 6-tooth blade at pressure setting a 2 ½. About 1/8" in. is generally allowed for the kerf and 3/16 in. for camber on heavier sections. Flat patterns and formed molybdenum sheet sections can be cut on a band saw with a fine tooth blade at very high speeds.



Abrasive cut-off wheels may also be employed. Wheel recommendations are indicated under grinding. Flame cutting, on the other hand, produces a very irregular edge.

Slitting and shearing may be done at room temperature for gages up to 0.025 in; heavier sections should be heated to about 400/1000 F. Shearing is feasible on sections up to about 1/8-in. thickness or diameter. Shearing camber can normally be held to about 1/8 in. in five feet (on 3/32-in. sheet); heavier sections, however, will show considerable drag.

SAWING & SHEARING [cont.]

Sections over 3/8-in thickness or diameter should be edge machined on a shaper or milling machine rather than sheared. Machining should be done along the edge rather than across, and it may be desirable to hold between steel plates during machining to avoid chipping the edges.

TURNING

		Speed, sfpm	Feed, ipr	Depth of Cut, in.
Roughing	high-speed steel	45/75	0.008/0.020	0.125/0.25
	C2 carbide	175/600	0.003/0.015	0.050/0.125
Finishing	C2 carbide	400/600	0.005/0.010	0.003/0.015

Previous studies indicated that a positive side rake angle in the range of 20/25° was essential; and other recommendations included a lead angle of 0° or slightly positive, relief angles of 7°, nose radii of 0.031/0.062 in., and honing all edges of the cutting tool at approximately 45° to the rake angles to give a 0.003/0.005-in. flat on cutting edge. A tough grade of straight tungsten carbide was found best with feeds of 0.005/0.010 ipr, while a general-purpose, C2-carbide grade could be used successfully on lighter finishing operations to obtain longer tool life.

Chlorinated oil and sulfur-base cutting oil can be used. If lubricants are not used, tool wear will be excessive. Sulfur-base oils cannot be used for machining electronic parts.

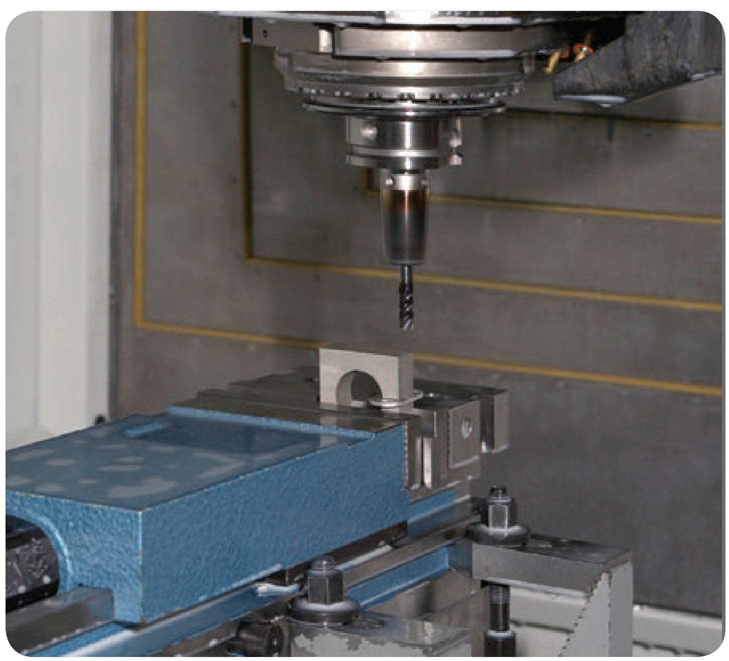
MILLING & SHAPING

		Speed, sfpm	Feed, ipr	Depth of Cut, in.
Rough Milling	C2 carbide	110/150	0.003/0.005*	0.050/0.010
Finish Milling	C2 carbide	300/400	0.003/0.005*	0.005/0.060
Shaping	C2 carbide	25/50	0.003/0.010	0.005/0.060

**per tooth chip load*

Milling and shaping are preferably done with C2 carbide grade tools of the design normally used for cast iron. Where production quantities make it desirable to use high-speed steel, shaping is preferred to milling, as sharper tools with a generous positive rake last longer and are easier to regrind.

Face milling is effective for machining plane surfaces on molybdenum parts. Face-milling cutters designed for machining cast iron with carbide tipped cutters are preferred, and soluble cutting fluids are essential for economic tool life.



DRILLING, REAMING & THREADING

		Speed, sfpm	Feed, ipr
Drilling	high-speed steel	25/150	0.003/0.005
	C2 carbide	40/175	0.003/0.005
Reaming	high-speed steel	15/20	0.005
	C2 carbide	20/30	0.003/0.007
Tapping	high-speed steel	15/20	–
	C2 carbide	20/30	–
Screw Cutting	high-speed steel	30/40	0.003/0.005
	C2 carbide	not recommended	

In drilling, two-lipped carbide drills are generally used. Cutting oil should be used for all drilling, reaming, tapping, or threading operations. When high-speed or carbide-tipped drills are used, the fact that molybdenum has a lower coefficient of expansion than steel makes it particularly important to keep the drills sharp and cool. It is worthwhile to regrind frequently to avoid difficulty and delay from binding. Special precautions are necessary with deep holes (more than three times the drill diameter) because of the abrasive molybdenum chips. These precautions may involve carbide wear strips along the shank, relieving the drill, feeding the drill from below or use of pressurized coolant.

Reaming is difficult, and tool life is very low compared to that obtained in machining heat-treated, low-alloy steel.

Threading can be done in various ways. Thread cutting with a single tool, grinding and roll threading are perhaps the most satisfactory. Die threading is not recommended, and tapping is not as easy as threading with a single-point tool. Coarse threads are preferred over fine threads, as very fine threads have a tendency to break. When coarse threads are developed, the depth of the thread needs to be only about 75% that normally cut in steel. For roll threading, the molybdenum should be heated to about 300 F.

SPECIAL MACHINING METHODS

Most special methods can be applied to molybdenum. Holes 1/8 in. ID by 12-in long, can be EDM machined, using brass electrodes and machining from both ends. Holes, 7 to 1000 microns in diameter, have been made in molybdenum by micromachining with "virtual electrode" in a 10% potassium-hydroxide electrolyte. Electron-beam machining is also applicable for holes in this size range.

GRINDING, BUFFING & HONING

Molybdenum is relatively easy to grind with conventional machinery and practices to any degree of finish and tolerance desired. It is important to use sharply dressed wheels with generous amounts of coolant since localized overheating can produce cracks in the surface of molybdenum. Soluble oils in emulsions of 1:40 to 1:60 are typically recommended.

The following grinding procedures are suggested as starting points; necessary changes, if any, will be evident from the results obtained in preliminary work.

Operation	Surface	Surface	Cylindrical	Cylindrical
type of grind	dry	wet	wet or dry	wet
wheel speed (surface fpm)	6000	6000	6500	6500
infeed (in./pass)	0.002	0.005	0.001	0.0005
work speed (surface fpm)	------	------	100	100
table speed (fpm)	50	50	1/3*	1/6*
crossfeed (in./stroke)	0.032	0.032	-----	-----
finish (microin., rms)	10	20	30	12

**width of wheel face/revolution of work*

Consult your local tool distributor for the most current grinding, buffing, and honing model available.
(Tool manufacturers include Iscar, Kennametal, Sandvik, etc.)

ELECTROPOLISHING & PHOTOETCHING

Molybdenum can be electropolished in a number of different solutions. Commercially the two most commonly used are phosphoric acid – sulfuric acid and straight sulfuric acid. The first solution requires a much higher current density than the latter but also gives a better finish. Both baths are used at room temperature with molybdenum as the anode.

Photoetching of molybdenum is readily done by conventional means. The unexposed portion is etched either chemically or electrolytically. It is possible to make parts too intricate or complicated for die stamping by this method with absolute uniformity of all parts and remarkably close tolerances. Generally photoetching is limited to sheet thickness from 0.001 to 0.010 in., with the minimum hole or mesh size never less than the thickness of the sheet.

FIRST SOLUTION 4 gal phosphoric acid 1 gal sulfuric acid 4 gal water	10/14 amp/sq.in.	dip in denatured alcohol prior to water rinsing
SECOND SOLUTION 2 parts sulfuric acid 1 part water	100/300 amp/sq ft	film of blue oxide formed, removed by immersion in alkaline cleaner or caustic-soda solution

Photoetching of molybdenum is readily done by conventional means. The unexposed portion is etched either chemically or electrolytically. It is possible to make parts too intricate or complicated for die stamping by this method with absolute uniformity of all parts and remarkably close tolerances. Generally photoetching is limited to sheet thickness from 0.001 to 0.010 in., with the minimum hole or mesh size never less than the thickness of the sheet.

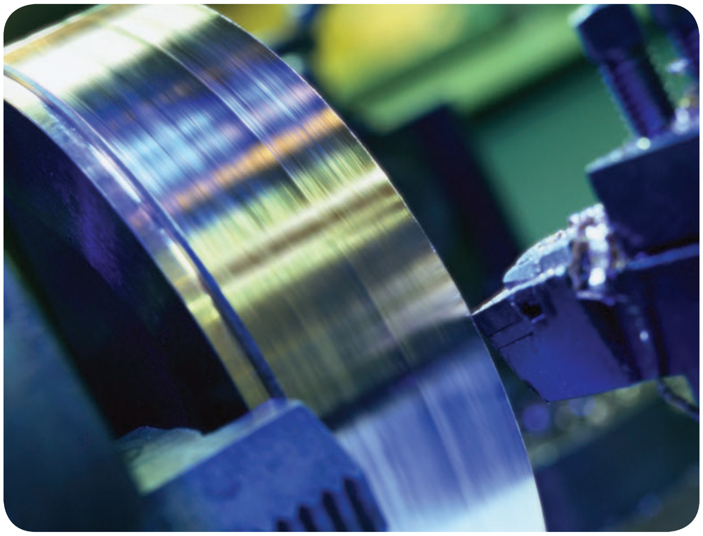
ED FAGAN INC.

Machining Guide for Tungsten Alloys

[as per ASTM B 777, EU RoHS Directive Compliant]

Tungsten Alloys* are alloyed with different elements such as nickel, copper, and iron to produce a large variety of grades. Many of these alloys have engineering properties similar to steel and are relatively easy to machine. These alloys can be drilled, milled, turned, and tapped using standard tools and equipment, while using speeds and feeds similar to Grey Cast Iron. Tungsten Alloys can also be plated or painted to enhance their corrosion protection.

Tungsten Alloys have found wide acceptance in applications such as radiation shielding, medical equipment, boring bars, vibration dampening, sporting goods, as well as counterweights in aircraft and racing cars.



PARAMETERS AND TOOLING TO ACCOMPLISH PROPER MACHINING:

TOOLS:

C-2 Grade carbide tooling is recommended. Use as generous a nose radius as possible.

OPERATIONS:

- Turning/Boring: Positive rake is recommended
  Roughing: 0.050"/ 0.200"
  Depth of Cut: 0.008"/.010" Feed.
- Finishing: 0.010"/0.030"
  Depth of Cut: 0.003"/0.005" Feed.
- Turning Speed: 250/350 Surface feet per minute.

Note: For above operations, air is the preferred method of cooling tools; coolant can be used.

DRILLING/TAPPING:

- Drilling: Use Carbide tipped or solid carbide drills with air or coolant such as Molydisulfide cutting fluid. Drill tap holes to 50-55% of thread hole requirement.
- Tapping: Use straight flute, high alloy taps. For small threaded holes, thread forming taps can be used. Nitrided or solid carbide taps will extend life of tap on long run jobs.

GRINDING:

Use Aluminum Oxide type wheels ("J" grade typical) with coolant to remove grinding material rapidly.

MILLING:

Feeds and Speeds should follow Grey Cast Iron recommendations.

Feeds: 0.003" per tooth as a starting point.

Speed: 75 - 750 sfpm with carbide tools;
adjust to depth of cut.

End Milling: Slight "climb" is best starting point.

Note: For above operations, air is the preferred method of cooling tools; coolant can be used.

EXOTIC OPERATIONS:

Wire EDM; Solid EDM; Waterjet Cutting can be performed on tungsten alloys.

* *Tungsten Alloys are also known as Mallory 1000, Densalloy, Fansteel 77 and Densimet.*

ED FAGAN INC.

General Guide to Machining Soft Magnetic Alloys

GENERAL

Soft Magnetic alloys can be machined by all of the common metal machining processes. No special equipment or procedures are required to produce parts with accurate dimensions with excellent finishes.

Soft Magnetic alloys that are primarily Nickel-Iron alloys (e.g. EFI Alloy 79, aka Magnifer 7904, HyMu 80, Hipernom, Moly-Permalloy and Permalloy 80, EFI Alloy 50, aka Magnifer 50, High Permeability 49 and Alloy 47-50 and Radiometal 4550) can be machined in accordance with the information found in our Machining Nickel-Iron Alloys available from our website.

This machining guide covers the machining of Hiperco 50, Hiperco 50A, and VIM VAR Core Iron.

HIPERCO 50/50A

TOOLS

The choice between high-speed steel and carbide tools depends largely on production quantities. When using carbide tools, surface speed feet/minute (SFPM) can be increased between 2 and 3 times over the high-speed suggestions, Feeds can be increased between 50 and 100%.

The following charts include typical machining parameters used to machine Hiperco 50A. The data should be used as a guide for initial machine setup only.

TURNING – SINGLE POINT & BOX TOOLS

	SFPM	IPR
High Speed Tools	30 - 40	.003 - .010
Carbide Tools	120 - 130	.005 - .010

DRILLING

Drill Diameter	SFPM	IPR
3/8″	30	.005
3/4″	30	.010

TURNING – CUT-OFF & FORM TOOLS

Cut-Off Tool Width	SFPM	IPR
1/16″	25	.001
1/8″	25	.002
1/4″	25	.003
Form Tool Width	SFPM	IPR
1/2″	25	.004
1″	25	.0025
1-1/2″	25	.002

REAMING

	SFPM	IPR
Under 1/2″	65	.005
Over 1/2″	65	.010

HIPERCO 50/50A *continued*

DIE THREADING

Threads per Inch	SFPM
3 – 7-1/2	8
8 – 15	10
Over 16	15

TAPPING

Threads per Inch	SFPM
3 – 7-1/2	6
8 – 15	7
16 – 24	11
Over 25	15

MILLING

SFPM	IPR
20 - 30	.001 - .005

BROACHING

	SFPM	IPR
Chip Load	8 - 15	.002
Over 25	15	

VIM VAR CORE IRON

The following charts include typical machining used to machine Electrical Iron. The data listed should be used as a guide for initial machine setup only.

TURNING

SINGLE POINT & BOX TOOLS

Depth of Cut, in.	High-Speed Tools			Carbide			
	Speed, ipm	Feed, ipr	Tool Material	Speed, ipm		Feed, ipr	Tool Material
				Brazed	Throw Away		
.150	80	.015	M-2	350	400	.020	C-6
.025	110	.007	M-3	400	490	.007	C-7

CUT-OFF & FORM TOOLS

Speed, fpm	Feed, ipr							Tool Material
	Cut-Off Tools Width, Inches			Form Tool Width, Inches				
	1/16	1/8	1/4	1/2	1	1-1/2	2	
70	.001	.0015	.002	.0015	.001	.001	.0007	M-2
250	.003	.0045	.006	.003	.0025	.0025	.0015	C-6

DRILLING

Speed, fpm	Feed, ipr								Tool Material
	Nominal Hole Diameter, Inches								
	1/16	1/8	1/4	1/2	3/4	1	1-1/2	2	
70	.001	.002	.004	.007	.010	.012	.015	.018	M-42

VIM VAR CORE IRON *continued*

TAPPING

Speed, fpm	Tool Material
15 - 20	M-1; M-7; M-10

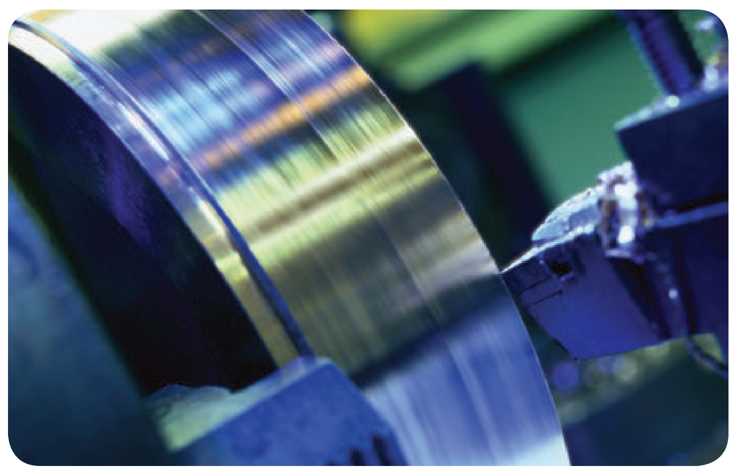
DIE THREADING

Speed, fpm				Tool Material
7 or less	8 to 15	16 to 24	25 and up, T.P.I.	
8 - 20	10 - 25	15 - 30	20 - 35	M-1; M-2; M-7; M-10

MILLING – END PERIPHERAL

Depth of Cut, In.	High Speed Tools						Carbide Tools					
	Speed, fpm	Feed - Inches per tooth Cutter Diameter, Inches				Tool Material	Speed, fpm	Feed - Inches per tooth Cutter Diameter, Inches				Tool Material
		1/4	1/2	3/4	1 - 2			1/4	1/2	3/4	1 - 2	
.050	60	.002	.003	.005	.006	M-42	300	.0025	.004	.006	.008	C-6

Figures used for all metal removal operations covered are average. On certain work, the nature of the part may require adjustment of speeds and feeds. Each job has to be developed for best production results with optimum tool life. Speeds or feeds should be increased or decreased in small steps.



HEAT TREATING FOR OPTIMAL MAGNETIC PROPERTIES

Items as supplied from the mill exhibit only a fraction of the soft magnetic properties which they are capable of attaining. To optimize their full magnetic properties, further heat treatment is a necessity.

Optimal heat treating procedures vary depending on the type of soft magnetic alloy – recommended heating temperatures, hold times, cooling rates, cycles, and types of atmospheres are all specific to each alloy. For the recommended heat treatment procedures for your material, please contact your Ed Fagan Inc. sales representative.

ED FAGAN INC.

Special Purpose Alloys

www.edfagan.com

EASTERN OFFICE

Ed Fagan Inc. 769 Susquehanna Avenue

Franklin Lakes, NJ 07417

Phone: 201-891-4003

Fax: 201-891-3207

East Coast Toll Free Service: 800-335-6827

WESTERN OFFICE

Ed Fagan Inc.

10537 Humbolt Street

Los Alamitos, CA 90720

Phone: 562-431-2568

Fax: 562-598-7122

West Coast Toll Free Service: 800-782-9657

EUROPE

Ed Fagan Europe Ltd.

Unit 3a South Hams Business Park

Churchstow, Kingsbridge Devon, UKTQ7 3QH

Phone: +44 (0) 1548 858 770

Fax: +44 (0) 1548 856 516

Email: sales@edfagan.co.uk

Website: www.edfagan.co.uk