General Guide to Machining Nickel-Iron Alloys

ED FAGAN INC.

769 Susquehanna Avenue, Franklin Lakes, NJ 07417 • ***phone* 201.891.4003**
10537 Humbolt Street, Los Alamitos, CA 90720 • ***phone* 562.431.2568**

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.

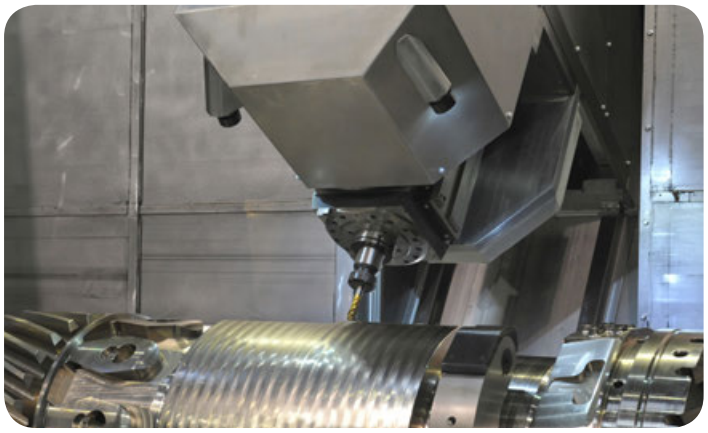
For machining with carbide tools, a K-6 manufactured by Kennemetal, Firthite HA manufactured by Firth Sterling, or #370 Carboloy could be used, or a K2S manufactured by Kennemetal, or a Firthite T-04 manufactured by Firth Sterling would be satisfactory. One thing of prime importance is that all feathered 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.*