GENERAL MACHINING PRACTICE
FOR CMI ELECTROMAGNETIC IRON

Electromagnetic Iron can be readily machined when proper tool angles are used. Tools should be ground to more acute cutting edge angles than are used for mild steel. The recommended angles and cutting conditions are given in Tables I and II. Since Electromagnetic Iron is a soft, ductile iron, it is difficult to push it off with tools ground with zero rake, as one would with carbon steel. With low rake angles, tremendous pressures can be generated, the tool may not want to feed properly, and the chips can be thick, blued, straight and stiff. On the other hand, with high rake angles, the tool cuts cleanly, the cutting pressures and heat generated are greatly reduced, the chips are bright and peel off in loose curls, and good surfaces are obtained. Electromagnetic Iron tends to machine more like copper than steel.

Because Electromagnetic Iron is quite ductile, the chips tend to form continuous curls rather than breaking up. For this reason, it may be necessary to shape the tool to divert the chips in the manner desired.

Good machining practices should be followed. For maximum rigidity, keep tools clamped tightly with as little overhang as possible. The cutting edges should be on center. It is important that drills, taps, cut-off tools and pick-ups be in good alignment. Use a copious supply of cutting fluid.

The data given here is offered as guides for proper tool design and application. For special conditions, it may be necessary to alter these designs. For example, when surface finish is secondary to cutting speed and stock removal, or when machining a hot-rolled round with mill scale still on, less rake combined with a coarser feed rate may be desirable.

It is advisable to provide side clearance on the tool to avoid rubbing and galling. In general, the clearances should be greater than for carbon steel.

MACHINABILITY RATING:
55 PERCENT OF B1112 BESSEMER SCREW STOCK

TOOL STEELS

The speeds suggested here are based on the use of high-speed tool steels of the M-42, T-5, T-15 or similar types. Higher speeds would be possible with carbide tool bits of the K-6, 883 or K-21 types. Since Electromagnetic Iron is soft, toughness rather than hardness is desired in the carbide tips. The back and side rake angles can be reduced somewhat when using carbide tool tips and higher speeds, but low rake angles should be avoided.

CUTTING FLUIDS

One of the principle functions of a cutting fluid is to cool the tool and the work piece. From this standpoint, a water-soluble oil is best and should be used whenever possible. Water-soluble oils now can be fully compounded with limited staining. If mineral oils are used, the viscosity should be as low as practical to provide maximum flow of oil to the cutting edge and conduction of heat away from the part. In either case, a chlorinated oil with active sulfur will provide maximum anti-weld protection. Some of the sulfurs become active only with the heat of the cutting tool. If sulfur is not permissible, use an oil with the next best anti-weld characteristics. Of course, sulfur should not be used with carbide tools.

In all machining operations, a flood of cutting fluid should be used if at all possible. Maximum production rates and tool life are dependent on adequate lubrication and cooling.

TURNING

Table I gives tool angles and operation conditions recommended for various operations using high-speed tool bits for finish turning. In some roughing operations, the feeds could be increased beyond those shown. Considerable leeway is permissible in speeds and feeds, but it is extremely important that the steep angles illustrated in Figures 1 and 2 be maintained.

Wide cut-off tools generate excess heat and remove more stock than necessary. They should be only wide enough to avoid lateral deflection. A lateral deflection can cause galling and rough surfaces as well as dishing. These deflections can be minimized by using low end cutting edge angles and by proper alignment. Blade tools provide better side clearance.

MILLING

Standard milling cutters can be used, but smoother finishes can be obtained by using cutters with a higher helix angle of from 45 deg. to 52 deg. and with front or radial rake angles increased to about 30 deg., as in Figure 3 and Table II. Cutting speeds can be 150 sfm or higher, depending upon the rigidity of the equipment. Table feed rates from ¾ to 1 ¼ inches per minute are recommended (or about 0.001 to 0.002 inch feed per tooth).

Milling saws can be of the standard hollow-ground, side tooth or stagger-tooth types. A front rake angle of 10 deg. to 12 deg., as in Figure 4, is available on standard cutters and performs satisfactory. However, steeper rake angles are available as a special and will give better performance. A side-tooth cutter, modified by chamfering the teeth alternately at 45 deg. for one-third their face width, as in Figure 5, has proved satisfactory. By using cutter speeds in the neighborhood of 300 sfm, advantage will be taken of centrifugal force to throw chips off the cutter and prevent them from being carried around into the next bite.

Feed rates will depend upon the finish desired and on the depth of the cut in relationship to the space provided between the cutter teeth. Regrinding, of course, reduces this space. The maximum feed rate will be that at which chips begin to load up between the teeth, and may be found to be in the range of 15 to 20 inches of table travel per minute (or about 0.001 to 0.002 inch feed per tooth.).

DRILLING

Drills can be standard, but better results will be obtained by either increasing the back or end clearance to 12 deg. or grinding a secondary clearance of about 20 deg. to provide greater accessibility of the coolant to the cutting edges. We do ***not*** recommended grinding on the face of the cutting edge, within the flute, to reduce the rake angle as is sometimes done for chip control. The same is true for the end mills and counterbores.

An advantage may be gained by decreasing the included angle of the cutting lips from 118 deg. to 100 deg. as in Figure 6. Various methods can be used for splitting or breaking the chips. The flute lips should be of equal length for minimum run-out.

The recommended cutting speed is 80 sfm, with light feeds of 0.002 to 0.004 inch per revolution.

TAPPING AND THREADING

Since Electromagnetic Iron is soft, ductile and flows readily, the best results are usually obtained with a form tap. With either a form tap or a cutting tap, it is recommended that drill size be larger than standard. This practice will reduce tap breakage and form a better thread. The tap tends to form a full thread because of the plastic flow of the ductile metal. For machine tapping, a two-flute spiral point may be advantageous.

When thread-tapping, a smoother and better thread will be formed with less effort if the tap is ground with a 10 deg. to 15 deg. radial rake or hook as in Figure 7. In all tapping operations, the hole should be free of all chips, to avoid tap breakage, and the work should be flooded with coolant.

When threading, use a 3- or 4-section chaser die with approximately 20 deg. radial rake or hook.

AUTOMATIC SCREW MACHINES

The above principles can be applied to automatic screw machines. The back and side rake angles can be adjusted to control the shape of the chips and their direction of travel, but acute angles should be maintained. Feed rates of 0.001 to 0.003 inch per revolution give best results.

Surface speeds of 150 feet per minute are permissible. Box tools are quite satisfactory for turning operations. Interrupted cuts can be used in long cuts if necessary for chip control. Form tools should be ground with rake angles of 20 deg. to 25 deg. if possible, as in Figure 8, to give good surfaces and reduce cutting pressures. The minimum rake angle would be 10 deg. to 20 deg. With zero rake angle, the tool will not want to cut, tool pressures will be high, and chatter may be produced. Front relief angles of 6 deg. to 8 deg. are suggested. The combination of high rake angles and sufficient front clearance will enable the feed to be heavy enough to keep the tool cutting smoothly and cleanly. The cutting edge should be on the center line of the work piece. For multiple diameters, some of the cutting edges will be slightly above or below the center line.

Blade tools have the advantage of greater ease in providing side clearance. In all cases, the tool should be well supported and rigidly clamped.

The use of a pick-up arm, properly aligned, will reduce to a minimum the amount of tip remaining on the part after cut-off.

In all screw machine operations, the cutters should be flooded with coolant.

TABLE I

ANGLES AND CONDITIONS FOR TURNING TOOLS

Tool Angles & Conditions	Single Point Turning	Cut-Off	Form Tool
Back Rake	15°	30°	20-25°
Side Rake	28-32°	–	–
End Relief or Clearance	8-10°	10-15°	6-8°
Side Relief or Clearance	8-10°	1-2°	1-2°
End Cutting Edge	15°	Up to 15°	–
Side Cutting Edge	15°	–	–
Nose Radius	1/32-1/16	–	–
Feed, inches/rev	.001-.003	.001-.003	.001-.002
Surface Speed, FPM	100-250	100-150	100-150

TABLE II

RECOMMENDED TOOLING AND CONDITIONS FOR MACHINING ELECTROMAGNETIC IRON

		Preferred	Acceptable
Milling Cutters:	Helix Angles, degrees	45-52	Standard
	Radial Rake, degrees	30	Standard
	Cutting Speed, sfm	150 or higher	
	Table Feed, in/min	3/4 to 1-1/4	
	Feed per tooth, inches	.001 to .002	
Milling Saws:	Types	Hollow ground Side-tooth Stagger-tooth	
	Radial Rake, degrees	15-20	10-12 Std
	Cutting Speed, sfm	300	
	Table Feed, in/min	15-20	
	Feed per tooth, inches	.001-.002	
Drills:	Included Lip Angle, degrees	100	118
	Front Rake	High helix	Standard
	Lip Clearance, degrees	12	8-12
	Secondary Clearance	20	
	Cutting Speed, sfm	80	
	Feeds, inches/rev	.002-.004	
Taps:	Types	Form	Cutting
	Radial Rake, degrees		12 to 15
	Cutting Speed, sfm	35	35

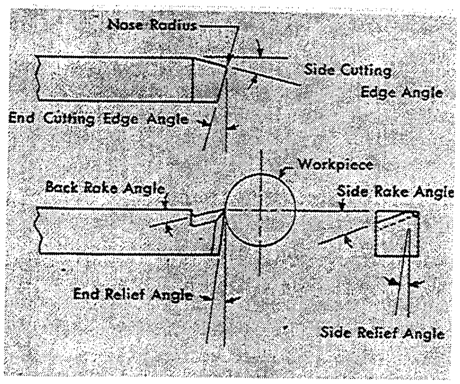

Figure 1 – High-Speed Turning Tool

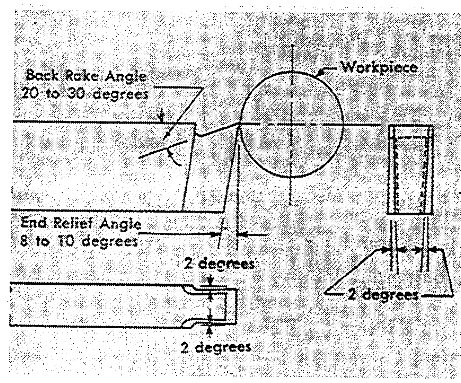

Figure 2 – High-Speed Cut-Off Tool

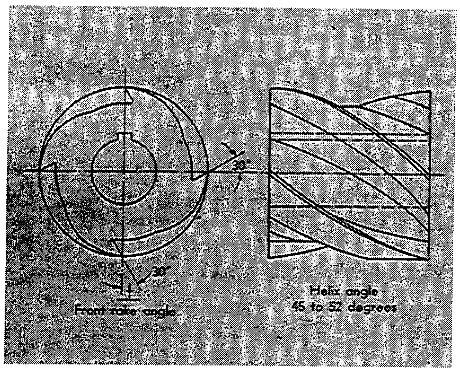

Figure 3 – High Helix Milling Cutter

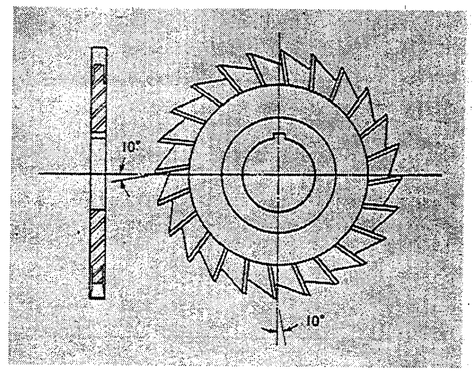

Figure 4 – Milling Saw

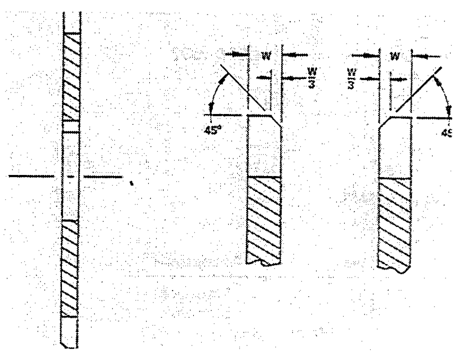

Figure 5 – Chamfered Milling Saw

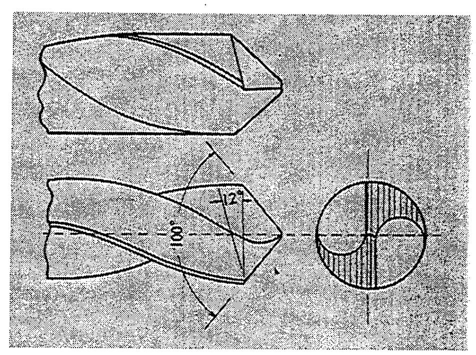

Figure 6 – Drill Point

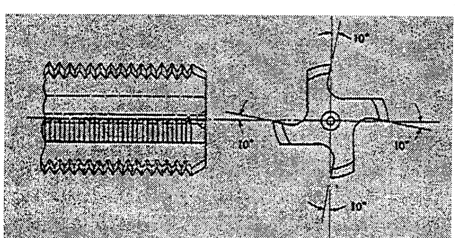

Figure 7 – Cutting Tap

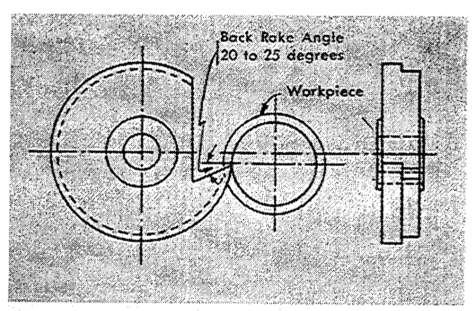

Figure 8 – High-Speed Form Tool