Engraving Tools and Cutters.
Spring Loaded Engraving Tools.
Vacuum Chucks for Workholding.

2L inc.
4 Kane Industrial Drive
Hudson, MA 01749
Tel: (978)567-8867, Fax: (978)562-8972

Engraving with a Rigid Tool

Engraving Tool Feeds and Speeds

Material	3000 RPM	6000 RPM	7500 RPM	10000 RPM	
Aluminum/Aluminum Alloys	6	12	15	20	
Brass/Bronze	6	12	15	20	
Copper/Copper Alloys	6	12	15	20	
Cast Iron, Soft	6	12	15	20	
Cast Iron, Hard	2.4	4.8	6	8	
Ductile Iron	3	6	7.5	10	
Malleable Iron	3	6	7.5	10	
Magnesium/Magnesium Alloys	6	12	15	20	Feedrates (IPM)
Monel/High Nickel Steel	3	6	7.5	10	
Nickel Base Hi-Temp. Alloys	2.4	4.8	6	8	
Plastics	9	18	22.5	30	
Plastics, Glass Filled	9	18	22.5	30	
Steel, Low Carbon	3	6	7.5	10	
Steel, Medium Carbon	4.5	9	11.25	15	
Steel, Hardened	1.5	3	3.75	5	
Stainless Steel, Soft	3	6	7.5	10	
Stainless Steel, Hard	1.5	3	3.75	5	
Titanium, Soft	3	6	7.5	10	
Titanium, Hard	1.5	3	3.75	5	
Copyright 2L inc. 2010, www.2Linc.com					

Please note as with all machining, it is important to take appropriate safety precautions.

RPM= Spindle Speed.
IPM = Inches per minute.

Plunge Feed to Depth at 50% of feedrates listed above. Typical engraving depths for permanently marking workpieces are 0.005” – 0.010”.

Notes: To reduce tip breakage on tougher materials or if small tip widths are being used, reduce feedrates above by 50% and make shallow passes (0.001" depth per pass or less is not uncommon on very tough materials). Use of the Tough Tip tool will allow faster feedrates at deeper cuts in tough materials.

Use of the Spring Loaded Engraving Tool will allow significant increase of feedrates and reductions in cycle times. At feedrates above 15 IPM ensure the high speed **lookahead feature** is enabled on the cnc machine to prevent rounding of corners and sudden starts and stops in corners.

Use of coolant will extend the life of the engraving toolbit.

Engraved lines that appear rough or jagged are usually caused by a dull toolbit or material buildup on the end of the toolbit. Material buildup mostly occurs with gummy materials such as aluminum or copper. Too fast a feedrate or too deep a cut may not allow the material to be cleanly cut. Taking a finishing pass of .001" - .002" deep at a slower feedrate can be used to reduce any burring or jagged edges. Using coolant to engrave helps reduce this buildup. Engraving Cutters are available specifically for softer materials.

Note: Variations in the above table may be required depending on material being engraved and cutting conditions. Consider the above recommendations as a starting point.

Engraving Tools and Cutters.
Spring Loaded Engraving Tools.
Vacuum Chucks for Workholding.

2L inc.
4 Kane Industrial Drive
Hudson, MA 01749
Tel: (978)567-8867, Fax: (978)562-8972

Engraving with a Spring Loaded Engraving Tool

Engraving Tool Feeds and Speeds

Material	3000 RPM	6000 RPM	7500 RPM	
Aluminum/Aluminum Alloys	15-25	30-50	37.5-62.5	
Brass/Bronze	15-25	30-50	37.5-62.5	
Copper/Copper Alloys	15-25	30-50	37.5-62.5	
Cast Iron, Soft	15	30	37.5	
Cast Iron, Hard	10	20	25	
Ductile Iron	13	26	32.5	
Malleable Iron	13	26	32.5	
Magnesium/Magnesium Alloys	20	40	50	
Monel/High Nickel Steel	13	26	32.5	
Nickel Base Hi-Temp. Alloys	13	26	32.5	Feedrates (IPM)
Plastics	25	50	62.5	
Plastics, Glass Filled	25	50	62.5	
Steel, Low Carbon	13	26	32.5	
Steel, Medium Carbon	13	26	32.5	
Steel, Hardened	10	20	25	
Stainless Steel, Soft	13	26	32.5	
Stainless Steel, Hard	13	26	32.5	
Titanium, Soft	13	26	32.5	
Titanium, Hard	10	20	25	
Glass & Stone (with diamond engraving tool - use coolant with spindle rotating or non-rotating)	17	Not Recommended		

Please note as with all machining, it is important to take appropriate safety precautions.

RPM= Spindle Speed.
IPM = Inches per minute.

- Plunge the tool to depth at a rapid feedrate. For example: G1Z-0.020F350 or G0Z-0.020
- Decrease the feedrate to produce deeper more pronounced marks and increase the feedrate to produce shallower less pronounced marks.
- When engraving tougher materials, use of the Tough Tip engraving toolbit is recommended.
- Ensure the high speed look ahead feature is enabled on the cnc machine to prevent rounding of corners due to the high feedrates required.

It has been observed that when using the latest generation of cnc machines, the fastest feedrate actually obtained when engraving letters 0.200" tall is approximately 80IPM. Increasing the spindle speed beyond 7500RPM requires feedrates that the cnc machines are not capable of providing and provides no additional benefit in cycle time reduction.

2L inc.
4 Kane Industrial Drive
Hudson, MA 01749
Tel: (978)567-8867, Fax: (978)562-8972

Spring Loaded Engraving Tool

Engraving Tool Feeds and Speeds – continued

- **The 3/4" Shaft Standard Spring Loaded Engraving Tool has been tested up to 10,000 RPM and has a 1,000 - 10,000 RPM recommended speed range.**
- **The 20 mm Shaft Standard Spring Loaded Engraving Tool has been tested up to 10,000 RPM and has a 1,000 - 10,000 RPM recommended speed range.**
- **The 1/2" Shaft Mini Spring Loaded Engraving Tool should not be used at speeds above 7,500 RPM and has a 1,000 - 7,500 RPM recommended speed range.**
- **The 3/8" Shaft Mini Spring Loaded Engraving Tool should not be used at speeds above 6,000 RPM and has a 1,000 - 6,000 RPM recommended speed range.**
- **The 1/4" Shaft Mini Spring Loaded Engraving Tool should not be used at speeds above 3,000 RPM and has a 1,000 - 3,000 RPM recommended speed range.**

(RPM speeds for the Mini Spring Loaded Engraving Tools are due to the reduced shaft diameter. Bending of the shaft may occur at higher RPMs).

Use of coolant will extend the life of the engraving toolbit.

Since the Spring Loaded Engraving Toolholder uses a spring to provide the downward pressure against the toolbit, in general, slower feedrates produce deeper more pronounced marks and faster feedrates produce shallower less pronounced marks.

Slow feedrates allow the spring to press the toolbit into the material being marked for a longer period of time and therefore produce a deeper mark. (i.e. if you completely stop feeding the tool, it will just drill down into the material until it is fully extended.)

Fast feedrates cause the toolbit to skim over the material being marked and produce a less pronounced mark. Too fast of a feedrate will cause the tool to skip over the material without cutting it and will produce marks that appear as dotted lines.

A depth controlling nosepiece is available for absolute depth control.

The feeds and speeds listed above are a good place to start and have been used successfully to engrave a wide variety of materials including aluminum, stainless steel, glass, and plastic parts. Increase or decrease the feedrates to achieve the desired results. Note that as you increase your spindle speed, your feed rates increase also. If you do not have a "look-ahead" feature on your machining center or CNC machine, fast feed rates may create a rounding affect on your engraved lettering.

The depth of cut (total amount of spring travel) can be up to .40". For standard engraving on flat surfaces, a depth of approximately Z-0.020 will compensate for normal material irregularities.

Note: Variations in the above table may be required depending on material being engraved and cutting conditions. Consider the above recommendations as a starting point.

Engraving Tools and Cutters.
Spring Loaded Engraving Tools.
Vacuum Chucks for Workholding.

2L inc.
4 Kane Industrial Drive
Hudson, MA 01749
Tel: (978)567-8867, Fax: (978)562-8972

Endmills

Feeds and Speeds

Material	SPEED [SFM]	D=1/8”	D=1/4”	D=1/2”
Aluminum/Aluminum Alloys	900-1500	0.0005	0.002	0.004
Brass/Bronze	400- 750	0.001	0.002	0.003
Copper/Copper Alloys	450- 850	0.001	0.002	0.002
Cast Iron, Soft	300- 750	0.001	0.002	0.003
Cast Iron, Hard	125- 350	0.0004	0.0008	0.002
Ductile Iron	150- 450	0.0005	0.001	0.002
Malleable Iron	300- 550	0.0005	0.001	0.003
Magnesium/Magnesium Alloys	1100-1600	0.001	0.002	0.004
Monel/High Nickel Steel	225-350	0.0005	0.001	0.002
Nickel Base Hi-Temp. Alloys	50-125	0.0004	0.0008	0.001
Plastics	900-1800	0.0015	0.003	0.006
Plastics, Glass Filled	400-1000	0.0015	0.003	0.004
Steel, Low Carbon	300- 600	0.0005	0.001	0.003
Steel, Medium Carbon	150- 450	0.0006	0.0015	0.002
Steel, Hardened	50- 225	0.0002	0.0005	0.001
Stainless Steel, Soft	250- 500	0.0005	0.001	0.002
Stainless Steel, Hard	75- 275	0.0002	0.0005	0.001
Titanium, Soft	175- 400	0.0005	0.001	0.002
Titanium, Hard	75- 250	0.0003	0.0005	0.001

Please note as with all machining, it is important to take appropriate safety precautions.

RPM = (SFM x 3.82) / D
IPM = (No. of teeth) x IPR x RPM

FOR SLOTTING APPLICATIONS: Reduce Speed to approximately 80% of the lower values for the particular material being machined.

AXIAL DEPTH OF CUT: These recommendations are for axial depths of cuts not to exceed 1 times the cutter diameter. On tougher materials, depths per pass as low as 0.001" are not uncommon.

Note: Variations in the above table may be required depending on material being machined and cutting conditions. Consider the above recommendations as a starting point.

RPM= Spindle Speed.
D = Diameter of tool.
IPR = Inches per rev.
IPM = Inches per minute.
IPT = Inches per tooth.
SFM and SFPM = Surface feet per minute.

Engraving Tools and Cutters.
Spring Loaded Engraving Tools.
Vacuum Chucks for Workholding.

2L inc.
4 Kane Industrial Drive
Hudson, MA 01749
Tel: (978)567-8867, Fax: (978)562-8972

Armor Mills

Feeds and Speeds

Material	Recommended Speed (SFPM) and Feedrates (IPT)	Cutting the below piece of granite, 2L used:	
		Cutter Diameter = .078" Depth of Cut per Pass = .020"	Cutter Diameter = .125" Depth of Cut per Pass = .030"
Granite Stone Armor Plate Hardened Steel	50- 125 SFPM 0.0001-0.001 IPT	6000 RPM F5.0 IPM Plunge = 2.0IPM	6000 RPM F6.0 IPM Plunge = 2.0IPM
Broken Taps and Drills	50- 125 SFPM 0.0001-0.0005 IPT	1500-2500 RPM Plunge = 0.1IPM	1500-2500 RPM Plunge = 0.1 IPM

Please note as with all machining, it is important to take appropriate safety precautions.

The sample of granite shown on our website was machined at the specific feeds and speeds listed above in the table. The Armor Mills successfully machined for 30 minutes at the above conditions to complete the sample.

Note: Consider the above feeds and speeds as a reference point. Variations in the above table may be required depending on material being machined and cutting conditions.

To remove broken taps and drills, it is recommended to feed the tools by hand (handwheel on lowest feedrate) until the tool has plunged far enough into the broken tool to encounter a stable cutting condition.

In other words, as the tool begins cutting into the uneven jagged surface of the broken tool, feed extremely slowly to prevent the tool from walking which will put high side loads on the cutting tool and lead to breakage.

Once the tool is fully into the cut and seems stable, feeding with the cnc control at the above feeds and speeds is possible.

Stones such as granite and marble have different machining properties among different varieties and even at different spots within the same piece of stone.

RPM = (SFPM x 3.82) / D
IPM = (No. of teeth) x IPT x RPM

RPM= Spindle Speed.
D = Diameter of tool.
IPR = Inches per rev.
IPM = Inches per minute.
IPT = Inches per tooth.
SFM and SFPM = Surface feet per minute.

Engraving Tools and Cutters.
Spring Loaded Engraving Tools.
Vacuum Chucks for Workholding.

2L inc.
4 Kane Industrial Drive
Hudson, MA 01749
Tel: (978)567-8867, Fax: (978)562-8972

Armor Drills

Feeds and Speeds

Material	Recommended Speed (SFPM) and Feedrates (IPT)	Cutter Diameter = .078”	Cutter Diameter = .125”
Granite Stone	50- 125 SFPM 0.0001-0.0005 IPT	1500-2500 RPM F5.0 IPM Plunge = 1.5 IPM	1500-2500 RPM F6.0 IPM Plunge = 1.5 IPM
Broken Taps and Drills, Armor Plate, Hardened Steel and Drills	50- 125 SFPM 0.0001-0.0005 IPT	1500-2500RPM Plunge = 0.1 IPM	1500-2500RPM Plunge = 0.1 IPM

Please note as with all machining, it is important to take appropriate safety precautions.

Note: Consider the above feeds and speeds as a reference point. Variations in the above table may be required depending on material being machined and cutting conditions.

To remove broken taps and drills, it is recommended to feed the tools by hand (handwheel on lowest feedrate) until the tool has plunged far enough into the broken tool to encounter a stable cutting condition.

In other words, as the tool begins cutting into the uneven jagged surface of the broken tool, feed extremely slowly to prevent the tool from walking which will put high side loads on the cutting tool and lead to breakage.

Once the tool is fully into the cut and seems stable, feeding with the cnc control at the above feeds and speeds is possible.

Stones such as granite and marble have different machining properties among different varieties and even at different spots within the same piece of stone.

RPM = (SFPM x 3.82) / D
IPM = (No. of teeth) x IPT x RPM

RPM= Spindle Speed.
D = Diameter of tool.
IPR = Inches per rev.
IPM = Inches per minute.
IPT = Inches per tooth.
SFM and SFPM = Surface feet per minute.