**Lubricants:**

Water soluble oil is the best.

**Sawing:**
Circular sawing - recommended blade is carbide tipped. Minimizes burring and requires less lubricant than high speed steel. Recommended blade design is a triple tooth with a 2 to 7 degree rake angle and deep gullets. Optimum sawing performance is obtained at 11,000 to 15,000 sfpm.

**Suggested feeds for use on general purpose machines:**

Feed per tooth (inches)

| Cutter Material | Face Mills | Helical Mills | Slotting Side Mills | End Mills | Form relieved | Circular saws |
|---|---|---|---|---|---|---|
| Carbon steel | .010 | .008 | .006 | .005 | .003 | .003 |
| High Speed Steel | .022 | .018 | .013 | .011 | .007 | .005 |
| Hard cast alloys/ Sintered carbide | .020 | .016 | .012 | .010 | .006 | .005 |

Feeds listed are a guide only. The feed used in the ultimate machining process will be determined by the surface finish required, the rigidity of the work piece and cutter, and the depth of the cut. Speeds of 15,000 sfpm for carbide cutters are not uncommon. High speed steel cutters are operated at speeds of up to 5,000 sfpm and high carbon steel cutters at speeds up to 600 sfpm.

**Milling:**

*Slotting* - recommended end mill: Two fluted high speed steel
Recommended operating speeds are from 360 to 475 sfpm. End mill can be operated without lubrication at low feed rates. Applying oil with a brush will improve surface finish.

**Fly cutting/Face milling:**

Recommended fly cutter or face mill: High speed steel.
Recommended lubricants - mineral or soluble oil

**Drilling:**
Recommended drills for cast plate should have more twists per inch than ordinary drills. Highly polished flutes. Use cutting compound.

*Shallow holes* - Depth should not exceed four times the diameter of the drill.
Recommended drill - Standard twist drills with conventional 118 degree point and 30 degree helix.

*Deep holes* - Depth exceeds four times the diameter of the drill
Recommended drill - High Felix (50 degree flute angle) drill with polished flutes and approximately 130 degree point angle.
For deep holes the feed rate should be reduced to about 50% of the maximum suggested rate to reduce heat buildup and chip packing in the flutes.

**Guidelines for Machining Aluminum Plate**

| Drill Size (inches) | Feed (ipr) | Speed (spfm) |
|---|---|---|
| 1/6 | .003 | 50 - 1,000 |
| 3/32 | .007 | 75 - 1,000 |
| 1/8 | .010 | 100 - 1,000 |
| 5/32 | .011 | 125 - 1,000 |
| 3/16 | .012 | 150 - 1,000 |
| 7/32 | .013 | 175 - 1,000 |
| 1/4 | .014 | 200 - 1,000 |
| 9/32 | .015 | 225 - 1,000 |
| 5/16 | .016 | 250 - 1,000 |
| 3/8 | .017 | 300 - 2,000 |
| 7/16 | .017 | 350 - 2,000 |
| 1/2 | .017 | 400 - 3,000 |
| 9/16 | .017 | 450 - 3,000 |
| 5/8 | .017 | 500 - 4,000 |
| 3/4 | .017 | 600 - 4,000 |

Reduce feed rate approximately 50% for all non heat treated aluminum products.

**Band Sawing:**

High speed saws with a blade speed of 3,000 - 6,000 feet per minute are recommended. Should be of tempered steel with a 15 degree minimum top rake and 4 to 8 teeth per inch.



Contact Clinton Aluminum & Stainless Steel – 800-826-3370

Clinton Aluminum & Stainless Steel