Con-Cast Zinc Alloy Bar Stock — Machining Guidelines

Zinc Alloys have good Machinability – providing high quality and productivity over a wide range of machining conditions. Long tool life, low cutting forces, good surface finish and small chip formation are easily achieved.

GENERAL RECOMMENDATIONS

Cutting Fluids	Cutting Tools	Speeds & Feeds
• Always apply Generously • Water-Based preferred, eg. 20:1 water-emulsifiable oil mixture.	• Large, polished flutes • Large rakes, clearances • Sharp cutting edges • Riged tool & work piece	• Moderate to high • Avoid dwells in feed

Note:

1.) The following recommendations were derived from studies using high-speed steel tools for machining ZA alloys in various cast forms. However they are also suitable for machining other zinc alloys castings, eg. Zamak alloys.

2.) The conditions presented are intended to give good overall results. If one aspect of machining requires optimization, other aspects may deteriorate. For example, optimum conditions for chip formation and cycle times (high speeds and feeds) are not the same as for optimizing surface finish, burr formation and tool life.

COMMON OPERATIONS

Drilling

Drill Design

- Fast spiral or polished general purpose jobber
- 118^0 point angle
- 12^0 to 14^0 lip relief angle
- DO NOT grind flats on lips

Speeds

- 200 to 300 ft/min (60 to 90 m/min)

Feeds

Diameter (in)	1/8	1/4	3/8	1/2	3/4
Feed Rate In/rev	0.004	0.008	0.011	0.013	0.016

Deep Hole (Depth > 6X Drill Dia.)

- Reduce feed and speed 50%
- Intermittent withdrawl

Tapping

Tap Design

- Spiral point for through holes
- Standard or special aluminum geometry for blind holes.

Speeds

- Up to 200 ft/min (60 m/min)
- Reduce speed as thread pitch decreases

Form Tapping

- Preferred for small diameters

Speed as for cutting tap

Sawing

Blade Design

- Common designs with raker set and standard tooth profile
- As coarse a blade as possible with at least 2 teeth in contact with workpiece.

Speeds

- 300 to 500 ft/min (90 to 120 m/min)

Feeds

- Vary with workpiece shape, blade geometry, surface finish requirements

Moderate feed initially and adjust as required

Milling

Cutter Design

- End Milling – Right hand spiral flute geometry and special aluminum cutter design recommended.
- Plain Milling – "High Helix" and "Helical Aluminum"

Milling Direction

- "Down" or "Climb" Milling

Depth of Cut

- 0.125 in (3.18 mm) for roughing passes
- 0.025 in (0.64 mm) or less for finishing passes

Speeds

- Up to 350 ft/min (100 m/min) for shallow depths of cut.
- Reduce speed for deep cuts, ie. As low as 100 ft/min (30 m/min)

Feeds

- High table feeds preferred – 0.005 to 0.015 in/tooth (0.13 to 0.38 mm/tooth).
- Optimum feed rate should be determined by trial – depends on cutter dia, width and depth of cut, number of teeth, cutter speed and machine and workpiece rigidity.

Reaming

Reamer Design

- Common 6-fluted reamer recommended
- Both right hand spiral fluted and straight-fluted geometries acceptable.

Hole Diameter

- Drilled hole approximately 97% of reamer diameter

Speeds & Feeds

- Similar to those for drilling

Reduce speed up to 50% for critical diameters

Turning

Tool Design

- 10 to 15^0 clearance angles
- 10 to 15^0 positive back and side rake angles

Speeds & Feeds

- 200 to 300 ft/min (60 to 90 m/min)

Moderate feed rates – 0.010 in/rev (0.25 mm/rev) and depths of cut – 0.1 to 0.15 in. (2.5 to 3.8 mm)

For Information Call
1-845-427-2151
Or email: rwinter@eazall.com

Eastern Alloys, Inc.
Henry Henning Drive - Maybrook, NY 12543
(845) 427-2151 FAX (845) 427-5185
www.eazall.com