

Machining • Grinding • Drilling Glass Sealing Alloys

29-17 F15 Alloy - 36, 42, 45, 46, 48, 49 & 52 Alloys

Machining

It is only possible to give some guidance on machining of (Kovar) ASTM F15 alloy.

Most users develop their own techniques and don't divulge the information

NOTE:

Cutting fluid selection is an important consideration of these types of alloys.

The presence of sulphur from the cutting fluid could result in serious degradation of the magnetic properties and could cause grain boundary embrittlement.

Animal lard is the suggested cutting fluid.

These alloys develop somewhat gummier and stringy chips. If sulphur is used in cutting fluid it must be completely removed prior to material being heat treated or going through any furnace operation.

Tool geometries normally used for austenitic stainless steel are applicable to these alloys.

Tools: Cobalt bearing tools suggested

If steel cutting tools are used, feed at approximately .010" to .012" per revolution at a speed as high as 35 feet per minute could be attained.

Tool Clearance

Side clearance angle	10 to 13 degrees
Front clearance angle	8 degrees
Back rake angle	12 to 14 degrees
Side rake angle	12 to 14 degrees

These angles are typical but vary considerably by the conditions of each individual heat that is melted.

Grinding

If grinding is uninterrupted the back rake should be zero to 4 degrees.

If a chip breaker is desired, it is suggested a step type be used about .010" deep and a width of about 3 times the feed. This can be done at about 90 feet per minute and possibly increased.

Stellite type 98 m-2 will allow 20% higher speed.

It is important that all feathered or wire edges be removed from the tools. Tools must be kept in top condition by repeated inspection.

Drilling

A 3/16" diameter hole speed of 40 feet per minute with the feed of .002 per revolution.

A 1/2" hole speed of 40 feet per minute with a feed of .004 per revolution.

Drills should be short and make a thin web at the point by conventional means.

Do not notch or make a crank shaft grind

Clean holes frequently to remove chips which will gall also keep the area cool.

Drill should be ground to a 118 degree to 120 degree included point angle.

Reaming speeds:

Reaming speeds should be about 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 deg. The tool should be as short as possible and have a slight face rake of about 5 deg. To 8 deg.

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 deg. To 10 deg. And the tap should have a .003" to .005" chamfered edge if possible. Concentric taps are considered best and should be as short as 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 electrolyzed tools be used. Speed should be about 20 feet per minute.