

## **GENERAL SET-UP INFORMATION**

- 1) Use end mill holder or heat or hydraulic shrink fit holders
- 2) Bury the tool in the holder as far as the application allows
- 3) Use two passes for coarse threads and one pass for fine threads
- 4) Use program where the linear feed is adjusted for internal or external threading
- 5) Start at the low end for the SFM and chipload per tooth
- 6) Put your tool diameter in the offset register on 0. Put in negative values to increase thread size
- 7) Make sure your start point is the center of the hole and the top of the part.
- 8) Make sure you call out your tool length offset at the top of the program
- 9) Make sure you have the correct D word for cutter compensation
- 10) Run the first part at 20-30% Feedrate override to remove keen edge and for safety sake.
- 11) Use coolant whenever possible.
- 12) First thread will normally come out a little undersize. Adjust offset and recut.
- 13) Try to use uncoated tools in aluminum. It builds up on the TIAIN coating.
- 14) After you get the thread to size, run until you have made two diameter offsets to get the size back. After these two offsets, pull the tool as the wear land will increase very quickly at this point.
- 15) Resharpening is available on these tools using any CNC regrind shop. It is economical on 3/8" shank and larger tools