CUSTOM POST PROCESSOR DEVELOPMENT

Thank you for your purchase of Mastercam Software. In an effort to make your software purchase more productive, we've included instructions on obtaining post processor support for your machine tools.

While many post processors have been provided on your included Mastercam Utility CD, it may be necessary to make minor changes to produce the NC code needed to drive your specific machine tool or to format the NC Code based on operator preferences.

Basic changes are included with your software purchase while on the most current revision of software as provided under the Mastercam Maintenance Program.

Please provide the following required information in electronic format so that we may process your change request:

- A copy of :
    - The Mastercam MCX file with completed toolpath operations that require changes.
    - Exiting Post Files, including the PST file located in your post processor directory. (i.e. C:\mcamX\mill\posts)
    - The existing control definition, including the control file located in the cnc_machines directory (i.e. C:\mcamx\cnc_machines\fadal.control)
- A modified NC program generated by the post processor provided, along with a mark-up file annotated to highlight the changes that are required. An example is provided below.
- Contact Name, Phone Number, Company, Machine Tool and Control, and the version of Mastercam.

A Zip2Go utility has been added to the Mastercam X interface that allows for these files to be gathered automatically by clicking on Help→Zip2Go Utility, then clicking the Create Zip2Go button located on the lower left corner of the Zip2Go form.

Please note that you will still need to provide the modified and tested file with your submittal. The name of the created file that you will need to send is located in the Blue windows bar at the top of the form.
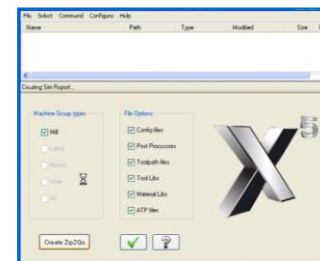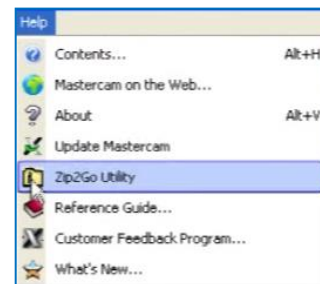
EXAMPLE OF NC CODE MARK-UP

ORIGINAL OUTPUT	REQUESTED OUTPUT
%	%
O0000	O0000
(PROGRAM NAME - T)	(PROGRAM NAME - T)
(DATE= 12-09-02 TIME= 08:06)	(Removed Date)
N100G20	N100 G20
N102G0G17G40G49G80G90	N102 G0 G17 G40 G49 G80 G90
(1/2 FLAT ENDMILL - 1 DIA. OFF. - 1 LEN. - 1 DIA. - .5)	N104 T1 (Remove tool comment replace w/tool number)
N104T1M6	N106 M6 (Add spaces for easier readability)
N106G0G90X-.5Y1.75A0.S1069M3	N108 S1069 M3 (Put spindle speed on separate line)
N108Z1.5	N110 G0 G90 X-.5 Y1.75 Z1.5 (Put Z value on first move)

Please email requests and all associated reference files contact information to posts@axsysinc.com. Please be sure to include your contact information.

If you have any additional questions, please contact our Technical Support Department at 248.926.8810 for further assistance.