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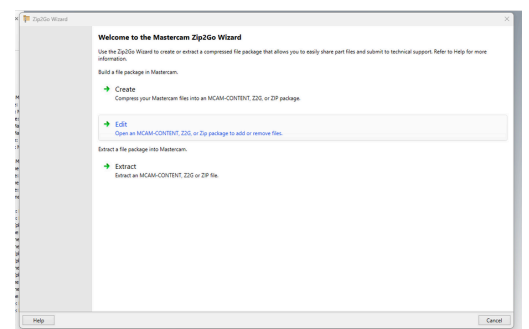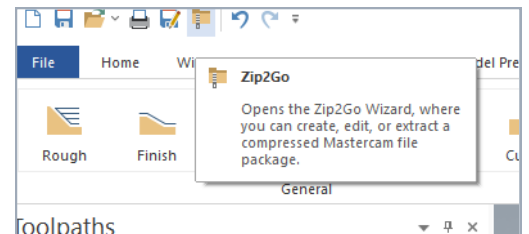
While basic post processors have been provided on your included Mastercam Utility CD, it may be necessary to make minor changes to produce the NC code needed to drive your specific machine tool or to format the NC Code based on operator preferences.

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 MCAM file with completed toolpath operations that require changes.
  - Existing post files, including the PST file located in your post processor directory. (i.e. C:\Users\Public\Documents\Shared Mastercam 2025\mill\Posts)
  - The existing machine and control definition files, located in the CNC_MACHINES directory (i.e. C:\Users\Public\Documents\Shared Mastercam 2025\CNC_MACHINES)

- **A modified NC program generated by the post processor provided, along with a marked-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 interface that allows for these files to be gathered automatically by clicking on File ⟶ Zip2Go.
Please note that you will still need to provide the modified and tested file with your submittal.