

Question/Issue

Control Definition: How do I change the default path for my NC files?

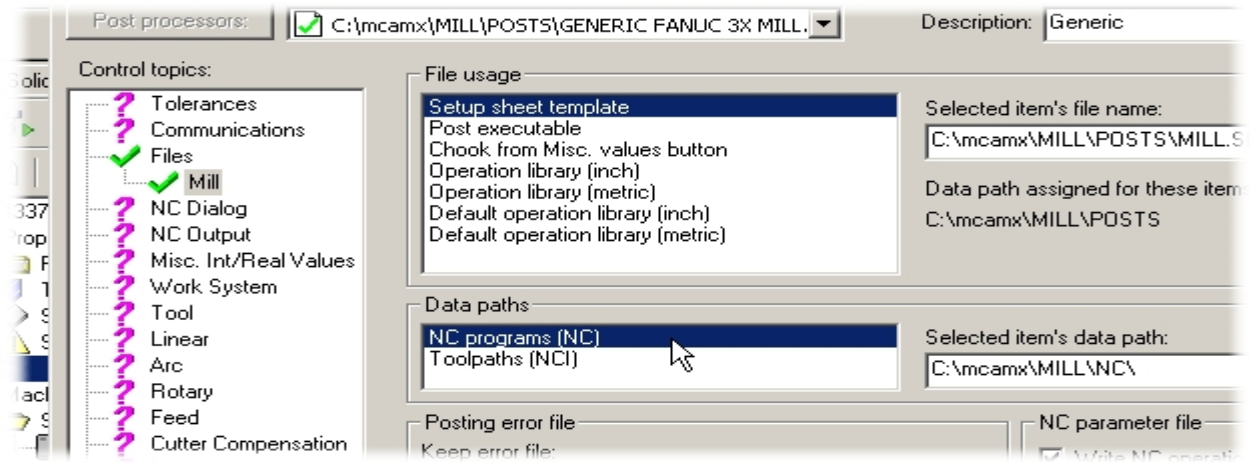
Answer/Solution

In Mastercam X2, the **Data paths** you define in the **Settings, Configuration, Files** properties page for Mastercam Mill, Router, Lathe, and Wire NC programs are used as defaults when you create Control Definitions.

Use one of the following methods to change a machine group's NC file path.

Changing the NC file path using the Control Definition Manager

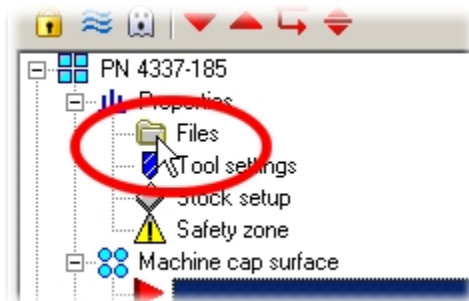
In the **Control Definition Manager, Files** page, you can set separate **Data paths** for NC and NCI files.



Changing the NC file path using the Machine Group Properties

Use the Toolpath Manager to change the NC file path for a selected machine group:

- First, open the **Machine Group Properties** dialog box, **Files** tab:



- Then type the new NC file path in the **Toolpath** field. You can also use options in the **Output comments to NC file** section at the bottom of this page to control which comments are processed by the post and are added to the NC file.

