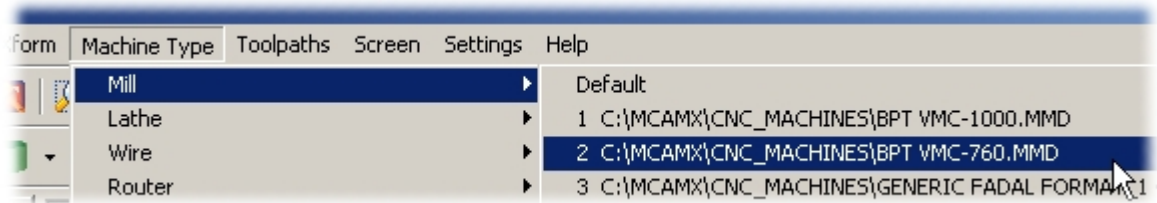


Question/Issue

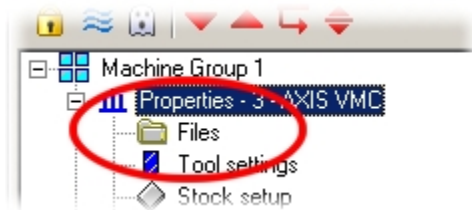
Post Processor: How do I select a different post processor?

Answer/Solution

For most users, selecting a post processor in Mastercam X is similar to Version 9. The major difference is that instead of selecting the .pst file directly, you need to select the machine definition that has been configured for the post processor. You can do this before creating your toolpaths:



or after your toolpaths have been created, through the machine group properties in the **Toolpath Manager**:

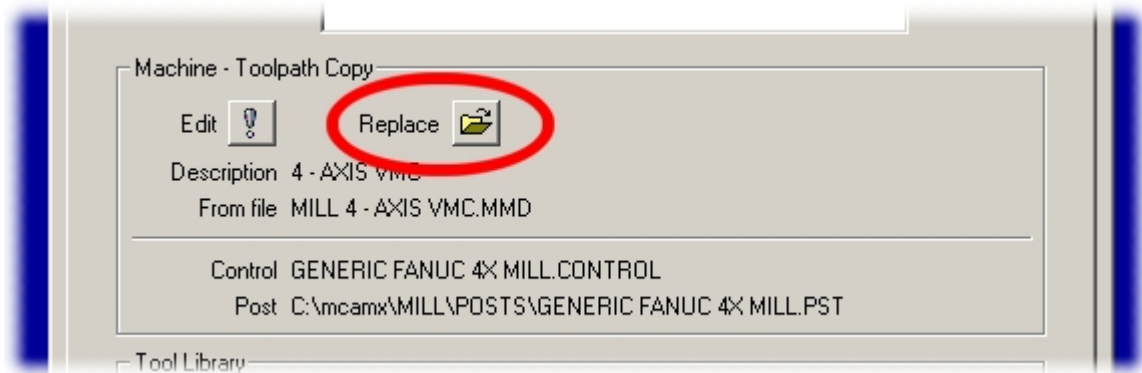


The [following procedure](#) shows how to select a different post processor for toolpaths that have already been created. Some shops might be configured so that each machine definition has multiple posts. In that case, follow [these steps](#) instead.

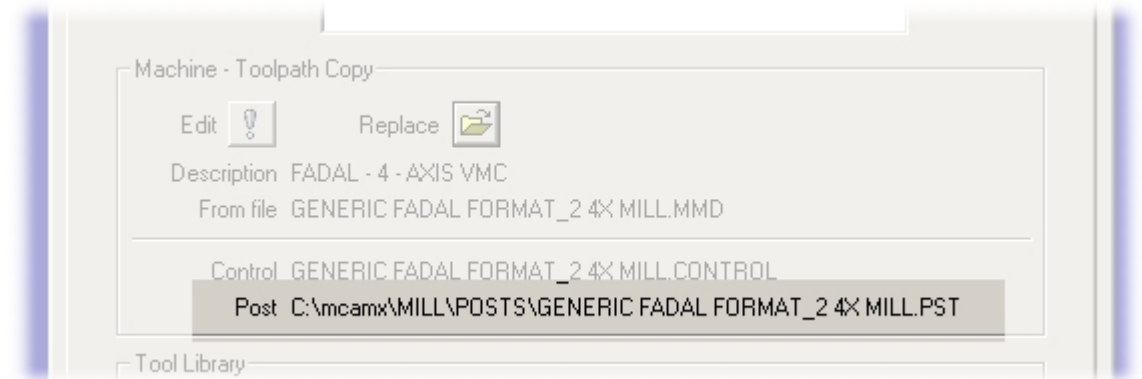
Selecting a different post processor This procedure applies to typical users whose machine definitions are set up to allow only a single control definition and post processor. In this case, you will select the new post processor by loading a different machine definition.

1. In the **Toolpath Manager**, click the **Files** icon in the machine group.

2. Click the **Replace** button in the **Machine** section.



3. Select the machine definition which contains the desired post.
4. Mastercam displays the name of the new machine definition with the new post processor.



5. Click **OK** to close the Machine Group Properties dialog box.