

Post Edits for Dummies!

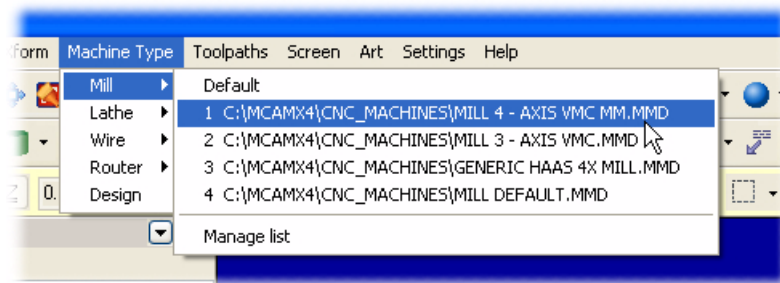
This article shows you how to make many common post edits—without actually needing to edit your post! The secret is that Mastercam’s control definition now lets you configure many common post output settings right in Mastercam—settings that used to require editing the .PST file.

How do I...?

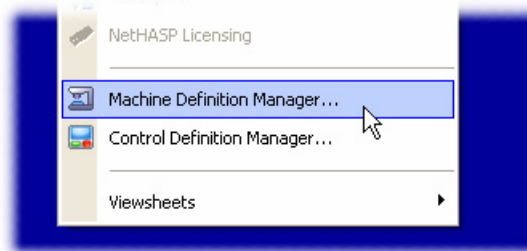
Open the proper control definition

Before you can make any of these edits, you need to open the proper control definition. The best way to do this is to follow these steps:

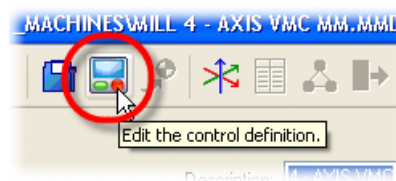
- 1 Select and load your machine definition like you normally do.



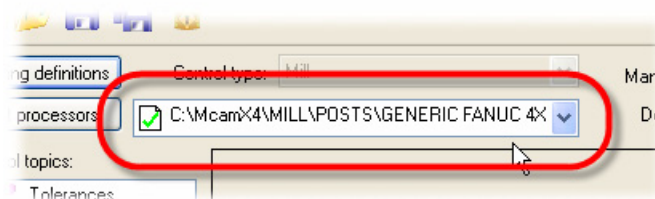
- 2 Select **Machine Definition Manager** from the **Settings** menu.



- 3 Click the **Edit control definition** button on the toolbar to open the **Control Definition Manager**.



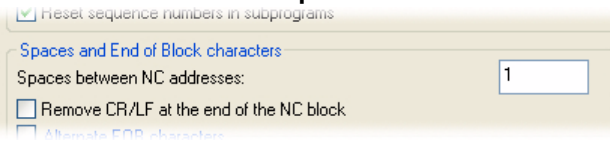
- 4 Make sure the name of your post is displayed with its proper location:



Control the number of spaces between words in the NC output

Follow these steps:

- 1 Go to the **NC Output** page in the **Control Definition Manager**.
- 2 Enter the desired value in the **Spaces between NC addresses** field.

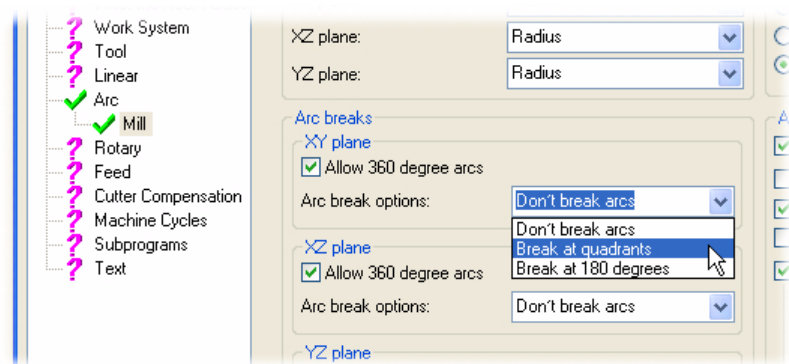


What this does in your post—This sets the **spaces\$** variable in your post.

Do not break arcs at the quadrants

Follow these steps:

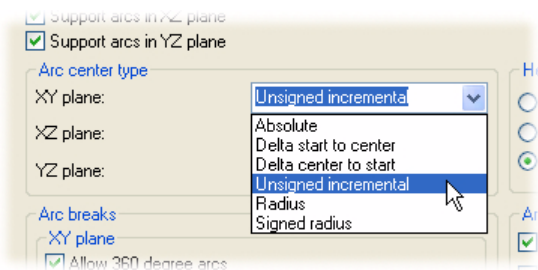
- 1 Go to the **Arc** page in the **Control Definition Manager**.
- 2 Select the desired options. You can set arc break options individually for each plane.



Output I,J,K with arcs

Follow these steps:

- 1 Go to the **Arc** page in the **Control Definition Manager**.
- 2 Select the **Arc center type**.



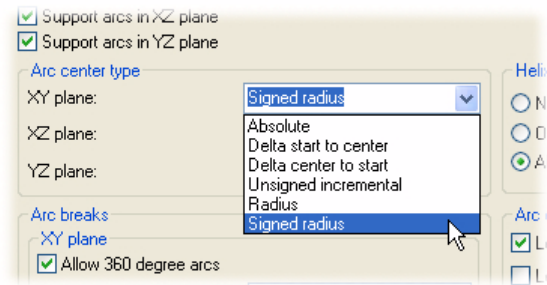
Three of the options will generate IJK output: **Delta start to center** and **Delta center to start** produce signed incremental output, or choose **Unsigned incremental**. You can set the arc center type individually for each plane.

What this does in your post—This selection sets the **arctype\$** variable in your post.

Output R signed when the sweep exceeds 180 degrees

Follow these steps:

- 1 Go to the **Arc** page in the **Control Definition Manager**.
- 2 Select **Signed radius** as the **Arc center type**. You can set the arc center type individually for each plane.

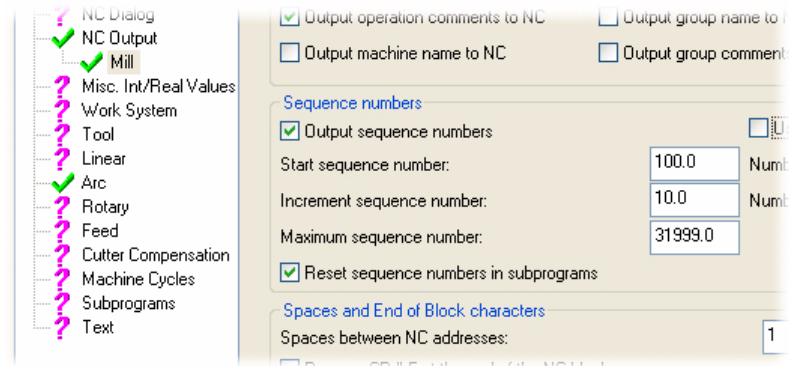


What this does in your post—This selection sets the **arctype\$** variable in your post.

Allow sequence numbers to be greater than N9999

Follow these steps:

- 1 Go to the **NC Output** page in the **Control Definition Manager**.
- 2 Enter the desired value in the **Maximum sequence number** field.

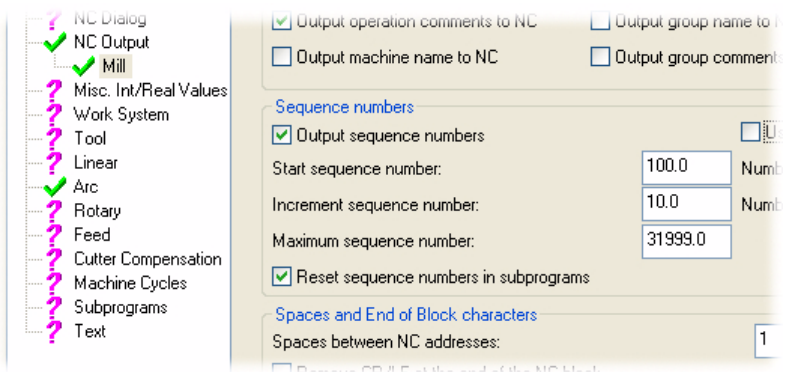


What this does in your post—This sets the **seqmax\$** variable in your post.

Omit sequence numbers entirely

Follow these steps:

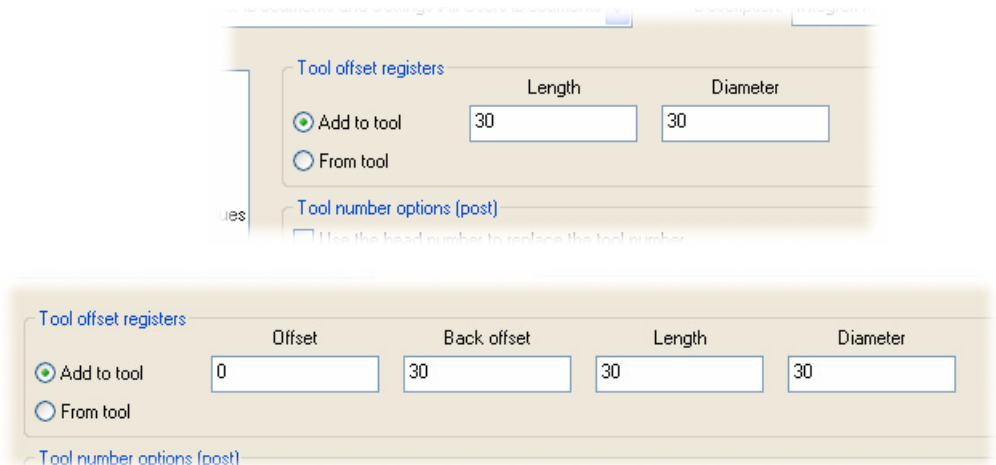
- 1 Go to the **NC Output** page in the **Control Definition Manager**.
- 2 Deselect the **Output sequence numbers** option.



What this does in your post—This sets the **omitseq\$** variable in your post.

Automatically number tool offsets by adding a number to the tool number

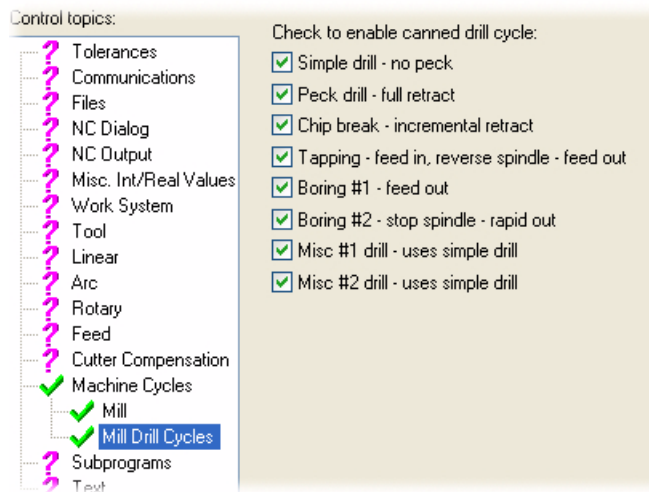
- 1 Go to the **Tool** page in the **Control Definition Manager**.
- 2 Select the **Add to tool** option.
- 3 Enter the desired values for each offset type. For example, if you want **T01** to use offset **H101**, enter **100** in the proper field. You can configure the numbering for each offset type individually. Different options are available for mill and lathe posts.



Select long code or canned cycle drilling output

Follow these steps:

- 1 Go to the **Mill Drill Cycles** page in the **Control Definition Manager**.
- 2 Select the cycles for which you want canned cycle output:



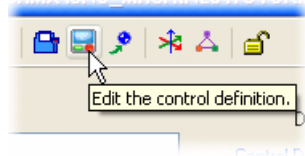
There are separate pages for mill and lathe canned cycles. If you are using a lathe or mill/turn machine, you will see the appropriate pages.

Use G92 instead of G54

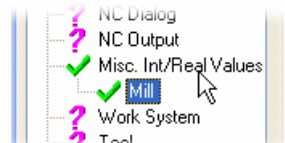
In Mastercam's generic posts, this is controlled by Miscellaneous Integer number 1. Use the **Control Definition Manager** to set a default value that is stored in the post text area of your post customization file. Operators can then choose to override this when creating individual toolpaths.

Follow these steps to set the default value.

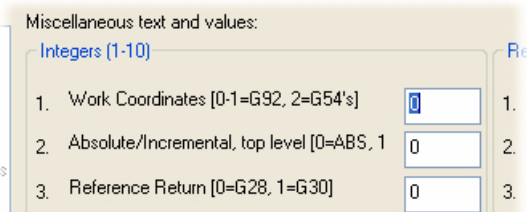
- 1 Select **Machine Definition Manager** from the **Settings** menu.
- 2 If necessary, select and load the machine definition that uses the desired post.
- 3 Click the **Edit control definition** button.



- 4 Select the **Misc Int/Real Values** page.



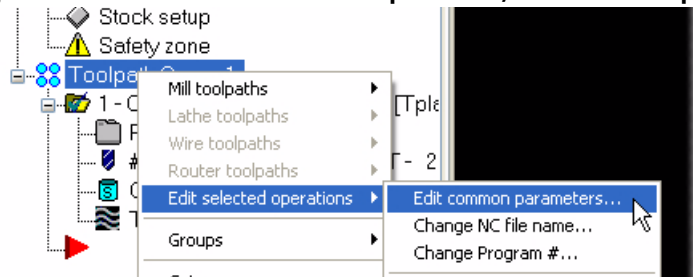
- 5 Change the **Work Coordinates** value to **0**.



- 6 Click **OK** to close the **Control Definition Manager**. Choose **Yes** when prompted to save the changes.
- 7 Click **OK** to close the **Machine Definition Manager**. Choose **Yes** when prompted to save the changes.
- 8 If you already have a machine group in the current part file that uses the control definition that you just edited, Mastercam prompts you to replace it with the edited version. Click **Yes**.

If your part file already has existing operations that use the edited machine definition, you will need to edit them from the Operations Manager in Mastercam.

- 1 Select the operations in the Toolpath Manager.
- 2 Right-click and select **Edit selected operations, Edit common parameters**.

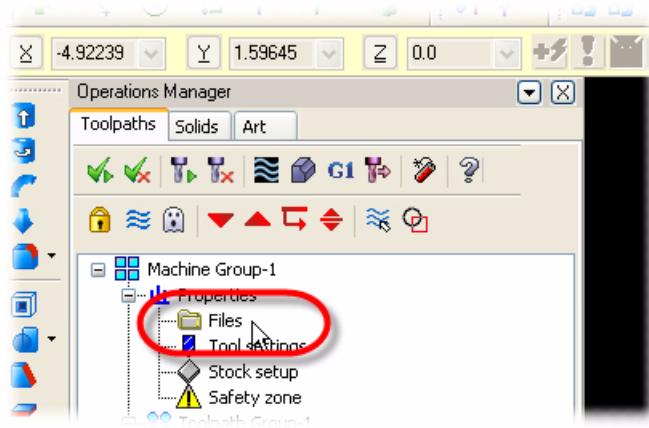


- 3 In the **Edit Common Parameters** dialog box, select the **Misc. values** button and change the **Work Coordinates** value to **0**.

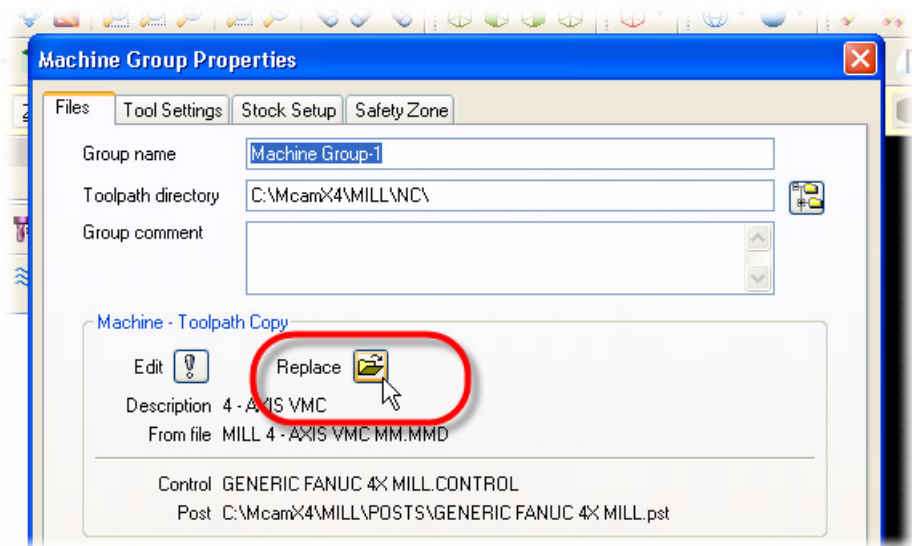
Make sure that my existing parts use the new settings

When you make changes to your control definition, you need to reload it into existing parts. Follow these steps before you post a part file if you want to use your new settings:

- 1 In the **Toolpath Manager**, click **Files**.



- 2 Click the **Replace** button.



- 3 Select the same machine definition from the **File Open** dialog box. This will reload the latest version of the machine definition, which should have your latest changes to the control definition.

Learn more about working with machine and control definitions

Go to our website and check out the “Working with Machine and Control Definitions” video lessons. There is also a PDF companion document. Click here: <http://www.mastercam.com/Multimedia/Default.aspx>