POST PROCESSOR INSTALLATION FOR MASTERCAM

STEP 1: Acquire the appropriate post processor from the following list:

> **techno6, techno7, techno8, techno9**
> Use with servo machines only!
> **mptechno**
> Simplified post processor. Use with stepper or servo machines.
> **mptechno9**
> The same mptechno post processor. Updated for Version 9.
> t**echnoX.zip**
> Post processor for McamX Router.

STEP 2: Copy the post to the appropriate location:
> c:\mill6\
> c:\entry6\
> c:\Mcam8\Router\posts
> c:\Mcam8\Mill\posts

STEP 3: Restart Mastercam.

FOR MILL6

> **-** Click on **Main Menu>Screen>Configure>NC Settings**
> **-** Set the Default Post Processor to the file you acquired.
> > NOTE: The button to the right will let you browse for the file.
> **-** Click on Done, Done and Yes.

FOR MILL OR ROUTER, V7-9

> **-** Click on **Main Menu>Screen>Configure>Files>File Usage>Post Processor.**
> **-** Below, where it says **File Name**, enter the file you acquired.
> > NOTE: The button to the right will let you browse for the file.
> **-** Click on OK and Yes.

FOR MCAMX

> **-** Unzip TechnoX.zip to a folder.
> **-** Move Techno.pst to C:\mcamx\router\posts
> **-** Move Techno_single_spindle.rmd and Techno.control to C:\mcamx\cnc_machines
> **-** Restart Mastercam
> **-** Click on Settings>Configuration>Files>FileUsage>RouterPostProcessor(PST).
> **-** Below, where it says "File Name", enter the file you acquired.
> > NOTE: The button to the right will let you browse for the file.
> **-** Click on OK and the check mark.
>
> Before creating a toolpath:
> **-** Click on **MachineType>Router>Select and chooseTechno_Single_Spindle.rmd.**

FOR MCAMX USING A DAVINCI MACHINE

> After installing the post:
> **-** Click on **Settings>Configuration>PostProcess>NC Extension**.
> **-** Enter **.NCD**