Installation and Updates
From Mastercam X, X2, X3 & X4 to
Mastercam X6

Getting your X6 installation up and running!
Installation

Inserting the DVD into your computer's drive, the Installation dialog will launch.
Installation choices are chosen. If options are required later, running the setup will allow the user to choose to install other options.
The current installation settings are shown. To change them, click the Configure Button.
Available settings Changes here once accepted, will update on the previous screen.
Acceptance of the license agreement. If the user clicks no, the software installation will end. You must accept the agreement for the installation to be completed.
Settings and License Agreement Once completed and accepted, the installation begins.
Installation – Complete

Select Exit to complete the installation.
Updating your Machines, Posts and Tools to Mastercam X6

This document will guide you through updating your Mastercam X, X2, X3 and X4 Machine Definitions, Control Definitions, Post Processors and Tools for use in Mastercam X6.

The first step is to gather the files that you need to update first and copy them to a location to update them from. It is suggested to create a new folder on your Desktop as shown below by right clicking on the Windows Desktop and selecting “New” then “Folder” on the sub menu. Type in a new name for the folder such as “X4toX6”.

Below is the new folder on the desktop.

If you are running several Mastercam products such as Mill, Lathe, Wire or Router, you may want to create separate temporary folders to update each product. Such as X4toX6-Mill, X4toX6-Lathe etc..
Next we need to copy the files that you USE on a daily basis to produce NC code. Show below is the directory structure for Mastercam X4 that has been installed in a MCAMX4 folder on the C:\ drive. The installed folder name you are updating from may be different. It could be MCAMX, MCAMX2 or MCAMX3.
In the image on the previous page the folder that we are most concerned with are boxed in. The “cnc_machines” machines folder is the most important and all Mastercam users will need to get files from that folder. The other folders will be used based on what Mastercam products you are running.

In this example we will be copying a single Mill Machine Definition and Control Definition. You will want to select all of the Machine Definitions and Control Definitions that you USE. In the table below show the file extensions, type and path for the file locations.

<table>
<thead>
<tr>
<th>File Extension</th>
<th>Description</th>
<th>Path</th>
</tr>
</thead>
<tbody>
<tr>
<td>*.CONTROL</td>
<td>Machine Control File</td>
<td>C:\McamX4\cnc_machines</td>
</tr>
<tr>
<td>*.MMD</td>
<td>Mill Machine Definition</td>
<td>C:\McamX4\cnc_machines</td>
</tr>
<tr>
<td>*.LMD</td>
<td>Lathe Machine Definition</td>
<td>C:\McamX4\cnc_machines</td>
</tr>
<tr>
<td>*.RMD</td>
<td>Router Machine Definition</td>
<td>C:\McamX4\cnc_machines</td>
</tr>
<tr>
<td>*.WMD</td>
<td>Wire Machine Definition</td>
<td>C:\McamX4\cnc_machines</td>
</tr>
</tbody>
</table>

Navigate to your “cnc_machines” folder and select the Machine & Control Definition files that you USE. You can “right-click” on your selected files and select “Copy”, as shown below.
Then go to the “X4toX6” folder on your Desktop and open the folder. Then “right-click” in the open area on the right and select “Paste” as shown below.

Next navigate to your “Posts” folder and select the *.PST files that you USE. You may also need to copy *.PSB files that most 5-Axis Mill Post and some Multi-Axis Mill/Turn posts use. You can “right-click” on your selected files and select “Copy” as previously described.

Then go to the “X4toX6” folder on your Desktop and open the folder. Then “right-click” in the open area on the right and select “Paste” as previously described.
Next navigate to your “Tools” folder and select the *.TOOLS files that you USE. You can “right-click” on your selected files and select “Copy” as previously described.

Then go to the “X4toX6” folder on your Desktop and open the folder. Then “right-click” in the open area on the right and select “Paste” as previously described.

Now open Mastercam X6.

Click on the “File” menu and select “Update Folder”
The “Update Folder” menu will display. To update your Machine & Control Definition set the upper portion of the dialog up as show below.

Be sure to set your “From this source folder:” to where you copied your Machine & Control Definitions. Set the “To this destination folder” to the correct location depending on what version of Windows® you are using:

For Windows XP® the correct directory would be:

For Windows Vista® & Windows 7® the correct directory would be:

Then click the to update your Machine & Control Definition’s.
Click on the “File” menu and select “Update Folder”. The “Update Folder” menu will display again. To update your Post Processors set the upper portion of the dialog up as show below.

Be sure to set your “From this source folder:” to where you copied your Post Processors. Set the “To this destination folder” to the correct location depending on what version of Windows® you are using:

For Windows XP® the correct directory would be:

For Windows Vista® & Windows 7® the correct directory would be:

Then click the to update your Post Processors.
Click on the “File” menu and select “Update Folder”. The “Update Folder” menu will display again. To update your Tool Libraries set the upper portion of the dialog up as shown below.

Be sure to set your “From this source folder:” to where you copied your Tool Libraries. Set the “To this destination folder” to the correct location depending on what version of Windows® you are using:

For Windows XP® the correct directory would be:

Then click the \[\] to update your Tool Libraries.
You may use the procedures above to update your MCX files, Operation Libraries, Default Files and Material files also.

Now that your files have been updated you can now set your default machines in Configuration and add your machines to the individual Machine Type lists.