Be sure you have the latest information!

Information might have been changed or added since this document was published. The latest version of this document is installed with Mastercam or can be obtained from your local Reseller. The ReadMe file (ReadMe.htm) includes the latest information about Mastercam features and enhancements.
## Contents

**Introduction** ................................................................. 1  
Before You Begin ............................................................... 1  
System Requirements ......................................................... 2  
If You Need More Help ....................................................... 3  
Additional Documentation ................................................. 4  
Mastercam Customer Feedback Program ......................... 5

**Copying Files** ........................................................................ 9  
Mastercam’s Folder Structure ............................................... 9  
  - Copy customized files to the User and Shared folders .......... 9  
Configuration Files ........................................................... 15  
  - Merge customized configuration files .............................. 15

**Converting Files** ............................................................ 19  
Update Folder ....................................................................... 19  
  - Convert Machine Definitions and Libraries ....................... 20  
  - Update customized configuration files ......................... 23  
Convert Parts ...................................................................... 27  
  - Convert parts with Update Folder ................................. 28

**Additional Procedures and Information** .......................... 31  
  - Update Custom Tool Profiles ......................................... 31  
  - Update ATP files ........................................................ 33  
  - Find and Fix Post Conversion Errors ............................ 33  
Agie Wire EDM Machines ..................................................... 34  
  - Edit control definitions for Agie Wire EDM machines ........ 35  
C-Hooks and Net-Hooks ....................................................... 37
INTRODUCTION

IMPORTANT: If you are updating to Mastercam X5 from Mastercam Version 9, see the Mastercam Version 9 to X5 Transition Guide, available from your Reseller.

This guide is designed to help Mastercam users re-create their Mastercam work environment—and their current jobs and parts—as quickly as possible after installing a new Mastercam release. It provides the information you need to update your work to Mastercam X5 from any previous Mastercam X version.

Recreating your Mastercam work environment in the new version involves several tasks:

- Copying files you have customized into the correct folders in the new installation.
- Updating files whose formats have changed in the new version.
- Modifying your configuration files so that they point to the correct files.
- Editing supplemental files such as custom tool profiles and posts to function correctly in the new version.

Before You Begin

Mastercam X5 presents major changes and improvements from earlier versions of Mastercam. This guide describes how to perform the transition to Mastercam X5 from any previous version of Mastercam X. To ensure the smoothest transition possible, follow the procedures in this guide as written and in order.

Note: In this guide, the term “Mastercam X” refers to any version of Mastercam X. When the information applies only to a specific version, the version number is indicated (for example, Mastercam X2 MR2).
## System Requirements

### Mastercam X System Requirements

| **Processor** | 2.5 GHz minimum (recommended)  
| | 32-bit or 64-bit Intel®-compatible processor  
| | Note: The processor must be SSE2-compatible. Intel-compatible processors manufactured since 2003 should be SSE2-compatible. |

| **Operating System** |  
| | - Windows XP, Windows Vista® (Business or Ultimate), or Windows 7 (Professional or Ultimate) including the latest service packs and recommended updates  
| | - VCRedist 2008 (Visual C++ Redistributable Package); installed with Mastercam |

| **Memory** | 2 GB minimum  
| | Additional RAM recommended for optimal performance |

| **Graphics Configuration** | 256MB OpenGL-compatible graphics card (minimum)  
| | - 1280 minimum width  
| | - 1280 x 1024 pixel resolution (minimum)  
| | - Full hardware acceleration display setting is recommended  
| | - Integrated graphics controllers are not supported |

| **Hard Disk Drive Space** | 3 GB available hard disk space (minimum) |

| **Mouse** | 2-button (minimum).  
| | 3-button or 2-button with middle mouse wheel (recommended). |
Mastercam X System Requirements

<table>
<thead>
<tr>
<th>Network Protocol</th>
<th>(NetHASP users only) TCP/IP protocol for NetHASP installation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Additional Supported Hardware</td>
<td>• Dual monitors.</td>
</tr>
<tr>
<td></td>
<td>• Multi-core processors. (One or more processor per instance of Mastercam.)</td>
</tr>
<tr>
<td></td>
<td>• SpaceBall® motion controllers.</td>
</tr>
<tr>
<td>Additional Software</td>
<td>• You will need the latest version of Adobe® Flash® to view the videos that accompany Mastercam.</td>
</tr>
<tr>
<td></td>
<td>• You will need Adobe® Reader® 6.0 (minimum) to display the PDF documentation.</td>
</tr>
<tr>
<td></td>
<td>You can download both applications from <a href="http://www.adobe.com">www.adobe.com</a>.</td>
</tr>
</tbody>
</table>

If You Need More Help

There are many ways to get help with Mastercam, including the following:

- **Mastercam Help**—Access Mastercam Help by selecting Help, Contents from Mastercam's menu bar or by pressing [Alt+H] on your keyboard. Also, most dialog boxes and ribbon bars feature a Help button (?) that opens Mastercam Help directly to related information.

- **Online help**—You can search for information or ask questions on the Mastercam Web forum, located at www.emastercam.com. You can also find a wealth of information, including many videos, at www.mastercam.com and www.mastercamedu.com.

- **Mastercam Reseller**—Your local Mastercam Reseller can help with most questions about Mastercam.

- **Technical Support**—CNC Software's Technical Support department (860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.

- **Documentation feedback**—For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

- **Mastercam University**—CNC Software sponsors Mastercam University, an affordable online learning platform that gives you 24/7 access to Mastercam training materials. Take
advantage of more than 180 videos to master your skills at your own pace and help prepare
yourself for Mastercam Certification. For more information on Mastercam University,
please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email
training@mastercam.com.

Additional Documentation

You can find more information on using Mastercam in the following materials, located in the
Documentation folder of your Mastercam installation:

- Mastercam X5 Installation Guide
- Mastercam X5 Administrator Guide
- Mastercam X5 Quick Start
- Mastercam X5 Reference Guide
- Mastercam X5 Transition Guide
- Mastercam X5 Quick Reference Card
- Mastercam X5 Wire Getting Started Guide
- Mastercam X5 NCI & Parameter Reference
- Mastercam X5 Post Debugger User’s Guide
Mastercam Customer Feedback Program

The Mastercam Customer Feedback Program is an optional data feedback tool. The program is completely voluntary. If you choose to turn this functionality on, it runs during every Mastercam session, automatically collecting and sending usage information to CNC Software, Inc. (CNC).

The data is used only for quality and product improvement purposes. It helps give CNC an idea of the relative value of various functionalities within its products. The data does not include user model data or any information that could be used to re-create models. This program cannot and will not collect data for the purpose of reverse engineering.

IMPORTANT: The Mastercam Customer Feedback Program will at no time collect data that is not relative to Mastercam. No sensitive, personal, or otherwise private data will be gathered.

The program is turned off by default. You must choose to participate in the program by clicking the option shown below in the Mastercam Customer Feedback Program dialog box.

This dialog box appears the first time you open Mastercam after an install. To participate, select the “Yes” option and click OK.
You can change your status at any time. To change your participation settings, select **Customer Feedback Program** from the **Help** menu.

The types of data that Mastercam collects includes the following:

- The Mastercam build number
- Your SIM number
- The name of the function you called and how you accessed it (toolbar button, hot keys, menu)

CNC combines the data gathered from Mastercam Customer Feedback Program participants to answer the following types of questions:

- Which Mastercam functions are used the most, and which are used the least?
- Which file translators are used, and how often?
- Which C-Hooks are used and how often?
- Which commands are used sequentially or in conjunction with each other?

*Note: The lists above are not comprehensive. These are examples of the types of data CNC collects.*

If you are running the Mastercam Customer Feedback Program, an icon (shown below) displays in your Windows program tray in the bottom right corner of your computer screen.
Notes:

- Participating in this program does not impact your processing time or computer performance.
- If you are running the Mastercam Customer Feedback Program, the tool automatically closes when you exit Mastercam.
- The data gathered by this program is sent to CNC when you exit Mastercam, or after a specific number of transactions.
- If you do not have an Internet connection, the data is saved to your local drive. The next time you start Mastercam, the program attempts to send the data. If the data cannot be sent over a period of five days, it is automatically purged from your system.
- You can turn your participation settings on and off at any time during a Mastercam session.
Copying Files

Mastercam’s Folder Structure

To be more compatible with Microsoft® guidelines and security measures, the default installation folder for the Mastercam application is now C:\Program Files\mcamx5 for 32-bit operating systems and C:\Program Files (x86)\mcamx5 for 64-bit operating systems. Additionally, this change requires the relocation of user-modifiable files from the Mastercam installation folder to two different locations, depending on whether the data is for a single user or if it is shared.

In previous versions of Mastercam, the installation folder was the default location of all user-modifiable files. Starting with Mastercam X5, these files fit into one of two categories: files that can be accessed and changed by a single user and files that can be accessed and changed by all users (shared files). Each category of customized file has its own storage folder: My mcamx5 or Shared mcamx5.

When moving data from a previous Mastercam version, you must place these files in their new locations.

Begin the transition to Mastercam X5 by working with customized files from previous versions that do not need conversion. In this section, you learn:

- The default locations and navigation shortcuts to Mastercam’s new storage folders
- The types of files that go into each new storage folder
- Which files can be copied to the new storage folders without conversion or updating.

Copy customized files to the User and Shared folders

Some customized files from previous versions of Mastercam X do not need conversion to the new X5 format. Just copy them to their new locations in the Mastercam X5 file structure. Most of these files, you can use “as is” in Mastercam X5. You will address the few that need minor modifications later.

1 Start Mastercam.
**TIPS:** If you are starting Mastercam X5 for the first time, two dialog boxes display information for the following:

- **Mastercam Customer Feedback Program:** The Mastercam Customer Feedback Program collects data while you use Mastercam, and automatically sends reliability and usage information to CNC Software, Inc. (CNC). Select *Yes, I would like to provide automatic feedback* to enroll. For more information, see page 5.

- **What’s New:** Information in the What’s New dialog box outlines the new features in each product level of Mastercam X5. It also includes demonstration videos and short tutorials to let you try some of the new functionality.

2 Choose **File, Open User Folder** from the menu bar to open the My mcamx5 folder.

The **My mcamX5** folder contains files that are customized for personal use. You can access or modify these files only when you log into your workstation. The files in this folder include:

- MCX, NC, and NCI files
- Personalized configuration files
- Other personalized files such as toolbars and key mapping

*Note: The My mcamx5 folder is not created until you run Mastercam X5.*
In the My mcamx5 folder, double-click the Shared mcamx5 shortcut to open the shared folder.

The Shared mcamX5 folder contains files that are shared by multiple Mastercam users. Any user who is logged into the workstation can access and modify them. The files in this folder include:

- Machine and control definitions
- Operation defaults files
- Posts
- Tool and material libraries
- Other shared files such as report templates, nesting defaults, and FBM configurations

The exact path of the My mcamx5 and Shared mcamx5 folders depends on your computer’s operating system. The following table lists the default paths for Windows XP, Windows Vista, and Windows 7.

<table>
<thead>
<tr>
<th>Operating System</th>
<th>Default Single User Path</th>
<th>Default Shared Path</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windows XP</td>
<td>C:\Documents and Settings&lt;Username&gt;\My Documents\my mcamx5</td>
<td>C:\Documents and Settings\All Users\Shared Documents\shared mcamx5</td>
</tr>
<tr>
<td>Windows Vista or 7</td>
<td>C:\Users&lt;Username&gt;\Documents\my mcamx5</td>
<td>C:\Users\Public\Documents\shared mcamx5</td>
</tr>
</tbody>
</table>

Mastercam provides three methods to make these two important folders easier to access.
From the File menu, select **Open User Folder** or **Open Shared Folder**.

- Double-click the **My mcaxm5** or the **Shared mcaxm5** shortcuts from within the My mcaxm5 and Shared mcaxm5 folders to open the folder in a separate window.
Select the **User Data** or **Shared Data** icons in the Places bar on the left side of the Open or Save As dialog boxes.

4 Locate the customizable files in your previous Mastercam version that do not need conversion.

5 Use Windows Explorer to copy these files to the appropriate X5 folders.

The following table lists the files from previous versions of Mastercam that can be used in Mastercam X5 without conversion and their default locations in both Mastercam X and Mastercam X5.
<table>
<thead>
<tr>
<th>File Type</th>
<th>Mastercam X</th>
<th>Mastercam X5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Report templates</td>
<td>.RPX</td>
<td>Used to create tool lists, part labels, nesting reports, and ActiveReports setup sheets.</td>
</tr>
<tr>
<td></td>
<td>location</td>
<td>\mcam\common\reports\ (MSC, NPL, NST, SST)</td>
</tr>
<tr>
<td>Nesting defaults</td>
<td>.DEFX .NSLX .NSTX</td>
<td></td>
</tr>
<tr>
<td></td>
<td>location</td>
<td>\mcam\NESTING</td>
</tr>
<tr>
<td>VB scripts</td>
<td>.VBS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>location</td>
<td>\mcam\vb</td>
</tr>
<tr>
<td>XML files</td>
<td>.XML</td>
<td>Examples include ATP levels and machine editors</td>
</tr>
<tr>
<td></td>
<td>multiple locations</td>
<td>\mcam\chooks\xatp\Levels</td>
</tr>
<tr>
<td></td>
<td></td>
<td>\mcam\common\editors\mastercam\Machines</td>
</tr>
<tr>
<td>HST Formula files</td>
<td>.FORMULA</td>
<td></td>
</tr>
<tr>
<td></td>
<td>location</td>
<td>\mcam\mill\formula</td>
</tr>
<tr>
<td>Art files</td>
<td>.ASL .BMP .MZX .ACS .APS</td>
<td></td>
</tr>
<tr>
<td></td>
<td>location</td>
<td>\mcam\art&lt;subfolders&gt;</td>
</tr>
<tr>
<td>NC and NCI files</td>
<td>.NC .NCI</td>
<td></td>
</tr>
<tr>
<td></td>
<td>multiple locations</td>
<td>\mcam&lt;mill, lathe, router, wire &gt;\nc</td>
</tr>
<tr>
<td>Configuration files</td>
<td>.CONFIG</td>
<td></td>
</tr>
<tr>
<td></td>
<td>location</td>
<td>\mcam\config</td>
</tr>
<tr>
<td>Key mapping files</td>
<td>.KMP</td>
<td></td>
</tr>
<tr>
<td></td>
<td>location</td>
<td>\mcam\config</td>
</tr>
<tr>
<td>Communications settings</td>
<td>.mcomm.CFG</td>
<td></td>
</tr>
<tr>
<td></td>
<td>location</td>
<td>\mcam\</td>
</tr>
<tr>
<td>NetHasp configuration</td>
<td>.nethasp.INI</td>
<td>Modification requires Administrator level permissions!</td>
</tr>
<tr>
<td></td>
<td>location</td>
<td>\mcam\</td>
</tr>
</tbody>
</table>
When you finish copying your customized files to their new X5 locations, close the three Windows Explorer windows.

### Configuration Files

**IMPORTANT:** Mastercam X5 introduces a major change to file locations. It is very important that the file paths in the configuration files reflect these changes. Re-creating your preferences “from scratch” from within the X5 environment using Mastercam’s default files (mcamx.config <English> and mcamxm.config <Metric>) ensures a smooth update. Using the Merge function is presented as an option for users that have heavily customized or numerous configuration files.

Your customized configuration files (.CONFIG) from previous versions of Mastercam X do not need conversion to work in Mastercam X5. However, you must modify them so that by the end of the transition, their settings will point to your converted files and libraries.

Although they are now located in ..\my mcamx5\CONFIG, some settings in the configuration files that you copied from previous versions of Mastercam X still point to files in your previous installation. In this section, you use the Merge function to overwrite obsolete data paths in these configuration files with the current version’s defaults. Merging disconnects your X5 configuration files from files in previous versions of Mastercam so that they cannot be accidentally overwritten. Later, after converting your files and libraries with Update Folder, you will update the settings to point to updated machine definitions and libraries that reside in the new X5 folder structure.

▶ **Merge customized configuration files**

Configuration files that you customized for Mastercam X can be used in Mastercam X5 with the following important qualifications:

- You must update default files and file paths in your configuration files to point to your converted libraries and files rather than the defaults. Do this only after you convert these files with Update Folder. Instructions for file conversion are included later in the Transition Guide.

- In Mastercam X5, some configuration property pages have been revised and some new pages have been added. Unless you customize these pages, Mastercam uses the default settings. Changes include:
  - **Chain Feature, Reports, Simulation,** and **Spin Controls** are new property pages.
  - Options on the **CAD Settings, Files, AutoSave/Backup, Screen, Solids, Toolpaths,** and **Verify Settings** property pages have been changed, added, or deleted.

You must manually configure these pages after you convert your files to X5 with Update Folder.

---

*Note: Skip the following section if you decide to create new configuration files, or if you wish to update the files and file paths manually.*
1. If you have minimized it, restore Mastercam.

2. Select **Settings, Configuration**. The System Configuration dialog box opens.

3. Select your custom configuration file from the **Current** configuration file drop-down.

4. Click the **Merge** button. The Configuration File Merge dialog box opens.

5. Click the **All Off** button. Mastercam clears the check boxes under “Sections to be merged.”
6 Under Sections to be merged, select the following options:
   - Default Machines
   - Files
   - Autosave/Backup
   - Start/Exit

Sections correspond to the property pages in the System Configuration dialog box. Each of the selected pages contains file path information that you will modify after you convert your libraries and machine definitions with Update Folder.

7 Choose a merge file type from the drop-down list.
   - Select Merge Initial Metric values from the drop-down list if you are merging a metric configuration.
   - Select Merge Initial Inch values from the drop-down list if you are merging an inch configuration.

8 Click OK.

Mastercam merges the factory defaults on the selected property pages into your customized configuration file. In the System configuration dialog box, a green check mark indicates each modified property page.

9 In the System Configuration dialog box, click OK.

10 Click Yes to save the changes to your configuration file

11 Repeat Steps 2 – 10 with your other customized configuration files.
Converting Files

Some files from previous versions of Mastercam X must be converted to the new X5 format before they can be used. CNC Software, Inc. understands that many current users have invested a substantial amount of time and energy in developing their tool and material libraries, toolpath and operation defaults, and operation libraries to meet their own preferences and customer requirements. To help make the transition process as smooth as possible, Mastercam X5 includes the Update Folder function to simplify the transfer and conversion of files from previous Mastercam X releases.

Update Folder

With Update Folder you can convert an entire folder of files to the current version of Mastercam. For example, you can convert all of your tool libraries and/or part file folders in a single operation. It does not matter which Mastercam product you are running or which machine definition is active.

As of Mastercam X5, a Mastercam version number is appended to most file extensions. In most instances, Mastercam file name extensions are formed by appending a hyphen followed by the Mastercam version. As part of the X5 conversion process, Update Folder changes the file extension of the files it updates.

The following table shows the files that Update Folder converts and their X5 file extensions. You should update these files with Update Folder before using them in Mastercam X5.

<table>
<thead>
<tr>
<th>File Type</th>
<th>Mastercam X Extension</th>
<th>Mastercam X5 Extension</th>
</tr>
</thead>
<tbody>
<tr>
<td>Part files</td>
<td>.MCX .EMCX</td>
<td>.MCX-5 .EMCX-5</td>
</tr>
<tr>
<td>Operation libraries</td>
<td>.OPERATIONS</td>
<td>OPERATIONS-5</td>
</tr>
<tr>
<td>Control definitions</td>
<td>.CONTROL</td>
<td>.CONTROL-5</td>
</tr>
<tr>
<td>Default files</td>
<td>.DEFAULTS</td>
<td>.DEFAULTS-5</td>
</tr>
<tr>
<td>Tool libraries</td>
<td>.TOOLS .HOLDERS .LHC .LIC</td>
<td>.TOOLS-5 .HOLDERS-5 .LHC .LIC</td>
</tr>
<tr>
<td>Machine definitions</td>
<td>.MMD .RMD .LMD .WMD .GMD</td>
<td>.MMD-5 .RMD-5 .LMD-5 .WMD-5 .GMD-5</td>
</tr>
<tr>
<td>Materials files</td>
<td>.MATERIALS</td>
<td>.MATERIALS-5</td>
</tr>
<tr>
<td>Wire power libraries</td>
<td>.POWER</td>
<td>.POWER</td>
</tr>
<tr>
<td>Post files</td>
<td>.PST .PSB</td>
<td>.PST .PSB</td>
</tr>
<tr>
<td>.SET Setup sheet files</td>
<td>.SET</td>
<td>.SET</td>
</tr>
</tbody>
</table>
Convert Machine Definitions and Libraries

The following procedure details an efficient way to convert Mastercam X files to Mastercam X5. For more information about using Update Folder, see Mastercam Help.

**IMPORTANT:** Only files supported by the products enabled on your SIM will be updated.

1. Select **File**, **Update Folder**.
   The Update Folder dialog box opens.
2. Choose **Update specific file types**.
3. In the **File types** box, choose all file types except **Part files**.

![Update Folder dialog box](image-url)
IMPORTANT: Converting part files with Update Folder can add significant processing time. We strongly recommend that you update parts to Mastercam X5 individually, by opening them with the File, Open function and saving them to the .MCX-5 file type.

If you decide to use Update Folder to convert parts in a batch, we recommend that you run the function first for your libraries and machine definitions as described in this section, and then again to convert your parts as described on page 28.

4 If your source folder contains files from multiple versions of Mastercam X, and you wish to update only files from specific versions to Mastercam X5, select Update specific versions to activate the Versions section. Then, select the desired versions.

5 In the From this source folder box, enter (if necessary) your previous Mastercam installation's root directory. (You can use the Browse button to navigate to the desired folder.)

6 If Search sub-folders isn't already active, select it.

7 Keep the default path shown in the To this destination folder box.

With the exception of part files, the file types converted by Update Folder are generally shared between multiple Mastercam users. By default, these converted files are copied to the Shared mcamx5 folder.

8 Click OK.

Mastercam converts all the files found in the source directory, including both metric and inch files.
Messages and files generated by Update Folder

As Update Folder processes your files, Mastercam might display the following messages and files.

- **Main level not visible.**
  Update Folder might generate this warning multiple times. Click OK to dismiss this message. Your data will not be affected.

- **Overwrite alert**
  Update Folder displays this message when it encounters another version of an updated file in the destination folder.
  - Click the second option to replace the original file with the converted one. Select **Apply this action to all subsequent conflicts** to continue processing without further interruption.

- **UpdatePost.log**
  When you select the **Post files** option in the File types section, Update Folder automatically runs the UpdatePost C-Hook and produces a log of the process, which displays in the Mastercam X Editor. The log file lists every error generated during the conversion, plus any
other instance where Mastercam could not unambiguously determine how to convert the original post.

- Print out or save the post log file to a different file name, and close the Mastercam X Editor. Use this log to troubleshoot errors in your posts.

Note: Each time you run Update Folder or UpdatePost and save posts to the same directory, the log is overwritten.

See page 33 for more information about correcting post errors.

▶ Update customized configuration files

After you convert the desired libraries and machine definitions to their Mastercam X5 locations, you can edit your configuration (.CONFIG) files to point to your converted files and libraries instead of to the factory defaults. Updating your configuration files after using Update Folder ensures that your defaults point to the appropriate X5 files.

If you merged your copied custom configuration according to the procedure described on page 15, you need to update only the property pages you selected in the Configuration File Merge dialog box (Default Machines, Files, AutoSave/Backup, Start/Exit). However, merging your custom configuration files overwrites all of the settings on these property pages not just the ones highlighted in this section. Review all of the settings on the property pages, and update as necessary.
Note: Skip this section if you want to use Mastercam’s default libraries and files for each product.

1. Select **Settings, Configuration**.
   The System Configuration dialog box opens.

2. Select your custom configuration file from the **Current** configuration file drop-down.

3. Select the **Default Machines** property page from the System Configuration dialog box topics list.

4. Set the default machine for any product by choosing an option from the drop-down lists.

   *Note: When a user selects the Machine Type, Default menu option, Mastercam uses the machines you select here to create the machine group.*
5. Select the **Files** property page from the topics list.

6. Modify the options in the **Data paths** and **File usage** sections to define default data paths for saving different file types.

**TIP:** While on the Files property page, you can also:
- configure Most Recently Used (MRU) drop-down menu settings
- define options for opening and saving files

7. Select the **AutoSave/Backup** property page from the topics list.

   The AutoSave/Backup property page is a subpage of the Files property page.
8 If you use AutoSave, review the file path in the **File name** field and modify it, if necessary.

9 Select the **Start/Exit** property page from the topics list.

10 Modify the **Startup settings** to reflect your file preferences for these options:

   - **Configuration**
   - **Toolbars**
   - **Keymap**

11 Click **OK**.
12 Click Yes to save the changes to your configuration file.
13 Repeat Steps 2 – 12 with your other customized configurations files.

**TIP:** In Mastercam X5, some configuration property pages have been revised and some new pages have been added. Unless you customize these pages, Mastercam uses the default settings. Changes include:

- *Chain Feature, Reports, Simulation, and Spin Controls* are new property pages.
- Options on the *CAD Settings, Files, AutoSave/Backup, Screen, Solids, Toolpaths,* and *Verify Settings* property pages have been changed, added, or deleted.

---

**Convert Parts**

Mastercam X5 converts your parts so that you can easily keep working on your current jobs. You can import individual files or use an import function (*File, Update folder*) that lets you convert parts in batch mode. However, many users find it more convenient to import parts one at a time, as they are required.

Importing parts from a previous Mastercam X version is straightforward, because Mastercam preserves all of the machine group and machine and control definition information.

- To convert individual parts, open them with the *File, Open* function and save them to the *.MCX-5* file type. (recommended)
- Use *File, Update folder* to convert parts in batch.

If you have many parts to convert, organize them so that the process is as efficient as possible. Update Folder converts all of the files in the source folder, so make sure it contains only the parts that you want to convert.

**IMPORTANT:** If you convert parts with toolpaths, it does not matter which product is running or what the active machine group is. However, if you convert parts that have geometry but no toolpaths, make sure that you are running in Design mode with no machine group visible in the Toolpath Manager.
Convert parts with Update Folder

**IMPORTANT:** Converting part files with Update Folder can take a significant amount of processing time. We *strongly recommend* that you update your parts to Mastercam X5 individually, by opening them with the File, Open function and saving them to the .MCX-5 file type. Back up your original files before using Update Folder to batch process parts.

1. Select **File, Update Folder**.

   The Update Folder dialog box opens.

2. Choose **Update specific file types**.
3. In the **File types** box, select only the **Part files** option.

![Update folder dialog box](image)

4. Select your desired source and destination folders.

**TIP:** Use the **Browse** button to navigate to the desired folders.

5. Deselect the **Search sub-folders** option.

![File types dialog box](image)

**Note:** Although Update Folder’s default destination folder is `\shared mcamx5\mcx`, Mastercam’s File, Open command goes by default to `\my mcamx5\mcx`. If you wish, you can change this setting on the **Files** property page in the Systems Configuration dialog box.

6. Click **OK**.

Mastercam converts all of your parts to the **.MCX-5** file format.
Additional Procedures and Information

If you have completed the procedures in the two previous sections, your transition to Mastercam X5 is nearly done. The following section details additional procedures and information that might be necessary to complete your transition.

▶ Update Custom Tool Profiles

Follow this procedure to edit custom tool definitions so that they reference the new Mastercam X5 file with the tool geometry.

*Note: This procedure assumes that you have already updated your tool libraries. (see page 20)*

1. Select **Toolpaths, Tool Manager**.
   
   You must have a machine type selected to open the Tool Manager.

2. Click the **Select a different tool library** icon, and open the custom tool library from the Select tool library window.

3. From the Select tool list window, choose **Library Tools** to display library tools only.

4. From the tool library window, select the tool you wish to update.
5 Right-click, and select **Edit tool** from the menu.

The Define Tool dialog box opens.

6 Click the **Parameters** tab.

7 Click the **Select** button to the right of the Tool file name field, and choose the part profile.

8 Click **OK** to close the Define Tool dialog box.

9 From the Tool Manager, click **OK**, and choose **Yes** to save your changes to the tool library.

To learn more about editing tool definitions, click the **Help** button in the Tool Manager dialog box.
Update ATP files

If you use the ATP C-hook, use Windows Explorer to copy your level files to the Levels folder. The X5 Levels folder is located in ..\Shared mcamx5\ATP. You can use Update Folder to convert your Operations libraries. (See page 19.)

You must manually re-create your strategies in Mastercam X5. See Mastercam Help for more information.

Find and Fix Post Conversion Errors

*Note: When you select the Post files option in the File types section, Update Folder automatically runs the UpdatePost utility.*

The UpdatePost utility incorporates several features that make it easy to identify and fix errors and areas of concern in the converted post. Because of the great variety of techniques used to customize Mastercam posts, it is impossible for UpdatePost to predict with 100% accuracy the intent of the post writer.

You need to be concerned about two types of situations:

- UpdatePost could not reliably convert a line or object in a way that conforms to Mastercam standards, and recorded an error condition instead of a complete line of code.
- There were several possible Mastercam conversion solutions, and UpdatePost was not sure which to use. This means that the converted post will work, but might not produce the output that you expect.

The most important tool in evaluating the conversion results is the UpdatePost log.

When you use Update Folder to update your posts, Mastercam writes this information to the file, UPDATEPOST.LOG, located in ..\Shared mcamx5\common.

**Tip:** Each time you run Update Folder or UpdatePost and save posts to the same directory, the log is overwritten. If you think you need to refer to the log later, make sure that you print it out or save a copy to a different file.

In the log file, an error looks like this:

22 - 20 Apr 2005 01:12:59 PM - PST LINE
(1200,1200) - Label has not been defined[*30]
The **LINE (x, y)** block tells you the location of the error in the original and converted post, in this case, line 1200 in each.

When you open the converted post, Mastercam displays the affected code as a block of three lines:

- The first line is the actual line of code created by UpdatePost.
- The second line is the original line, commented out, and preceded by the phrase `CNC<<CONVERT>>`.
- The third line reproduces the error message from the log, formatted as a comment, and preceded by the phrase `CNC<<MSG ... >>`.

The following code shows a typical conversion error. In this example, Mastercam knows that the line is supposed to be an `else` command, but it cannot convert the other word because it has not been properly defined:

```plaintext
else, # 'nobrk' emulation option
CNC<<CONVERT>> else, p_nobrk_emul # 'nobrk' emulation option
#CNC<<MSG-ERROR(1200,1200)>> Label has not been defined[*30]
```

There is a one-to-one correspondence between messages embedded in your code and messages in the log file. Before using a converted post, always review the log and the corresponding message in the post and make any necessary corrections.

See the Mastercam X5 Post Debugger User's Guide for more information.

**Agie Wire EDM Machines**

If you used a previous version of Mastercam Wire, you might have attached C-Hooks to the Aux Register button and the Misc Vals button on the Wire Parameters page. Converting your post processor with UpdatePost does not automatically attach these C-Hooks. These C-Hooks are installed in the `\chooks` directory of your Mastercam installation.

**IMPORTANT:** Mastercam X4 Wire enhanced support for Agie EDM with Agievision controls. The enhanced Agievision support includes a new interface and a complete set of machine definition, control definition, and post processor files.

The new Agievision interface is incompatible with existing parts that were created with the Agie C-Hook. The part geometry can be used, but all wirepaths must be re-created.

You can use the following procedure to update a control definition so that you can run and maintain existing Agie C-Hook parts, but you cannot use a control definition modified for the Agie C-Hook with the new Agievision interface.

See the Mastercam X5 Wire Getting Started Guide or the online help for more information.
**Edit control definitions for Agie Wire EDM machines**

1. From the **Settings** menu, choose **Control Definition Manager**, and open the desired control definition.

   Do not open the enhanced Agievision control definition. See the *Mastercam X5 Wire Getting Started Guide* or the online help for more information.

2. If necessary, select **Wire** from the **Control type** drop-down menu.

3. Under **Control topics**, choose the **Files** property page.

4. Under **File usage**, select **Chook** from **Misc. values button**.

5. Under **Select item's file name**, click the file cabinet button, navigate to the `PROGRAM FILES\MCAMX5\CHOOKS` directory, and select the `agie.dll` C-Hook.
Click **OK** to change the data path.

Select the option to **Enable chooks from misc values button**.

Selecting this option allows you to access the C-Hook's interface from the Wirepath dialog box.

Under **File usage**, select **Chook from Aux. register button**.

Under **Select item's file name**, click the file cabinet button, navigate to the `\chooks` directory, and select the `agiereg.dll` C-Hook.

Click **OK** to change the data path.

Select the option to **Enable chooks from aux registers button**.

Selecting this option allows you to access the C-Hook's interface from the Wirepath dialog box.

Click **OK** to close the Control Definition Manager.

Click **Yes** to save the edited control definition.
C-Hooks and Net-Hooks

C-Hooks (*.DLL) and NET-Hooks (*.NET) are add-in applications or utilities that customize, enhance, or extend Mastercam’s functionality. They can be created by individual users, Mastercam Resellers, third-party application developers, or by CNC Software itself. They must be modified prior to use with Mastercam X5, but they are not converted or updated by Update Folder. Please contact the application developer for more information.

C-Hooks and Net-Hooks are located in the Mastercam’s installation folder (..\Program Files\mcamx5\chooks). You must have read and write permissions to modify files in this folder.
Attention! Updates may be available. 
Go to Mastercam.com/Support for the latest downloads.