How to install a custom post processor in Mastercam 2019

This guide shows you how to install a custom post processor in Mastercam 2019 using a Simquest post processor package. You extract the zip file, copy the machine, control, and post files into the correct Mastercam folders, and add the new machine to your active machine list. When you finish, the machine appears in the Toolpaths Manager and is ready for programming.

Prerequisites

Before you begin, make sure you have:

- A Simquest post processor zip file (for example, CQMILL_FANUC.zip)
- Mastercam 2019 installed on your system
- An extraction tool, such as 7-Zip or Windows Extract

Step 1: Download the post processor zip file

Your new Simquest post processors arrive as a zip file. Download the file to a convenient location on your system, such as the Desktop or Downloads folder.

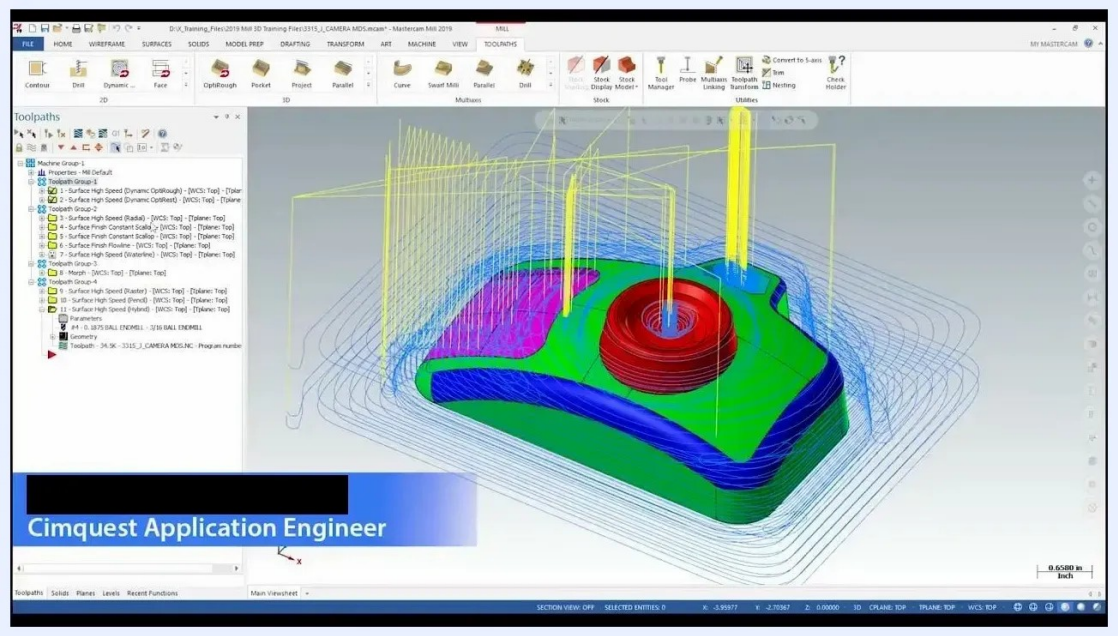
Step 2: Extract the files from the zip file

1. Locate the downloaded zip file (for example, CQMILL_FANUC.zip).
2. Right-click the zip file and select your extraction tool, such as 7-Zip or Windows Extract.
3. Choose **Extract to <folder name>** to create a new folder that contains all the files.
4. Open the new folder to view its contents.

The folder typically includes these files:

- A control definition file (for example, CQMILL_FANUC.mcam-control)
- A machine definition file (for example, CQMILL_VMC_FANUC.mcam-mmd)

- One or more post processor files (for example, CQMILL_FANUC.pst and CQMILL_FANUC.psb)

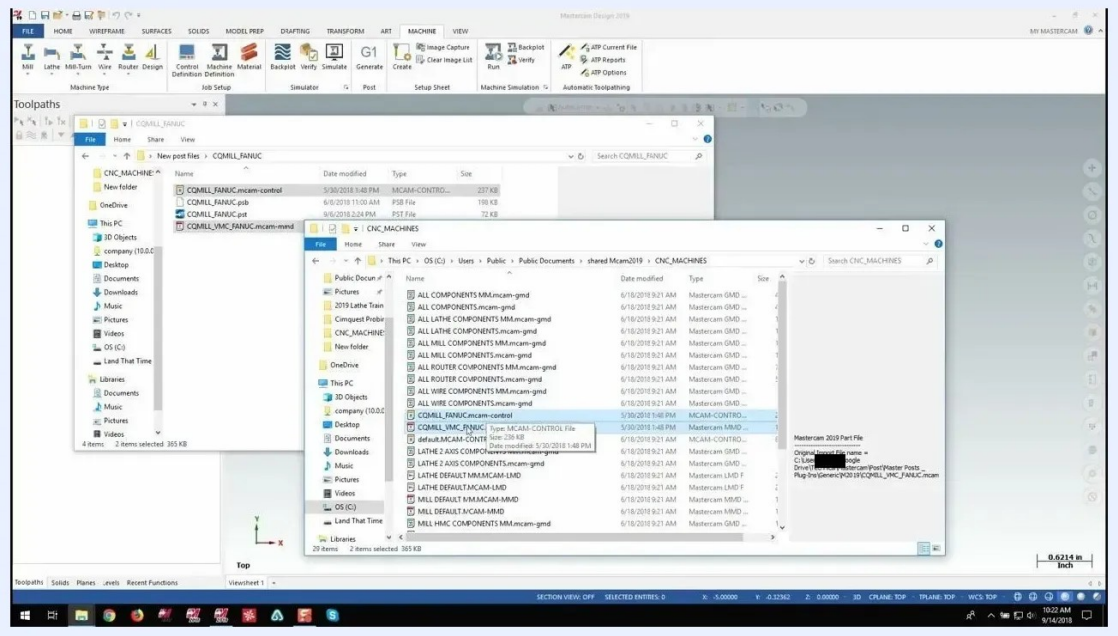


Windows File Explorer showing the right-click context menu on the Simquest zip file, with the extraction options visible

Step 3: Copy the machine and control definition files

1. In the extracted folder, select the machine definition file (.mcam-mmd) and the control definition file (.mcam-control).
2. Copy the files. Right-click and select **Copy**, or press Ctrl+C.
3. Navigate to C:\Users\Public\Documents\shared Mcam2019\CNC_MACHINES.

4. Paste the files into the CNC_MACHINES folder.



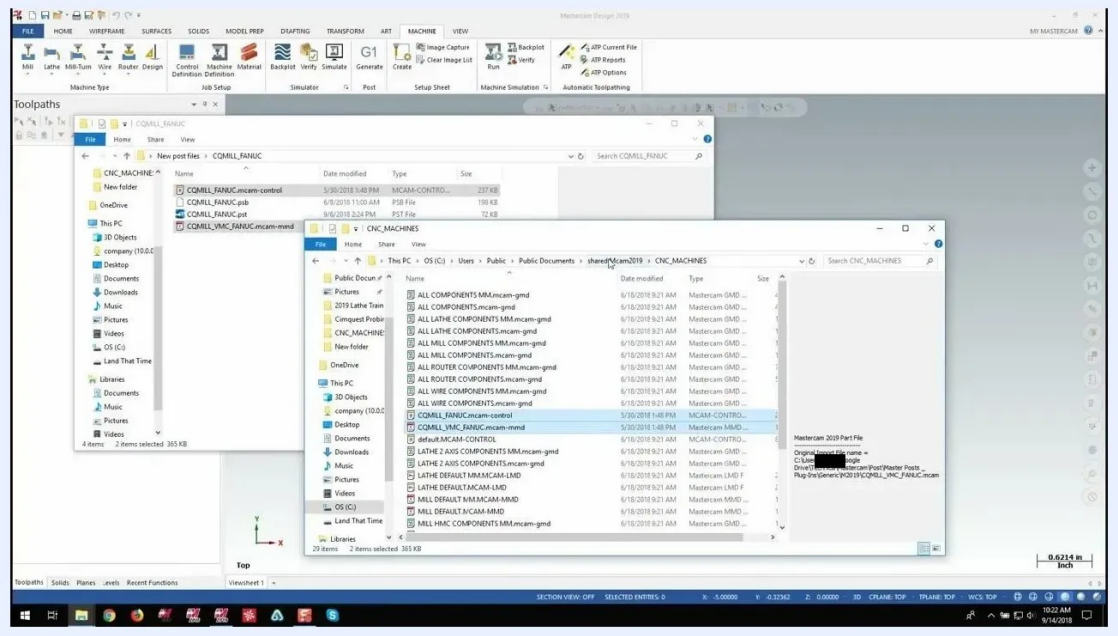
Windows File Explorer showing the CNC_MACHINES folder with the machine and control definition files being pasted

Step 4: Copy the post processor files to the Posts folder

Post processor files (.pst and .psb) must go in the subfolder that matches your machine type. This example uses a mill.

1. Navigate to C:\Users\Public\Documents\shared Mcam2019\mill\Posts.
2. Go back to the extracted folder and copy all the .pst and .psb files.

3. Paste them into the Posts folder.



Windows File Explorer open to the Posts folder for mill machines, with the context menu showing the Paste option

Step 5: Verify that all files are in place

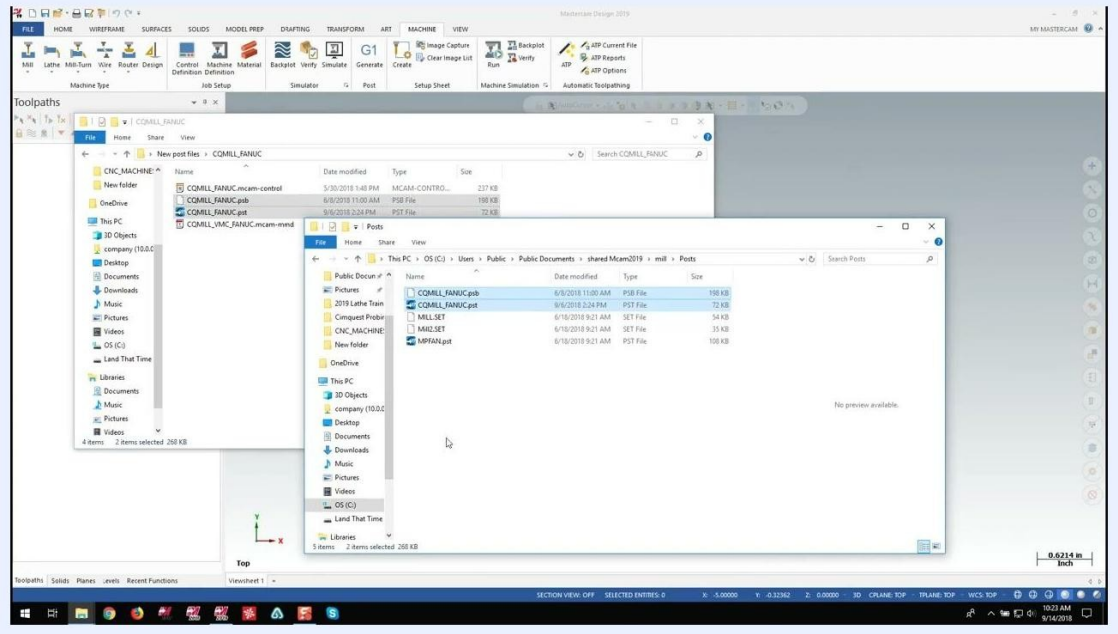
Confirm that the files are now in their respective folders:

- Machine and control definition files in CNC_MACHINES
- Post processor files in mill\Posts

Step 6: Open the Machine tab in Mastercam 2019

1. Launch Mastercam 2019.
2. Click the **MACHINE** tab in the ribbon at the top of the screen.

3. Make sure you are in the correct workspace for machine setup.



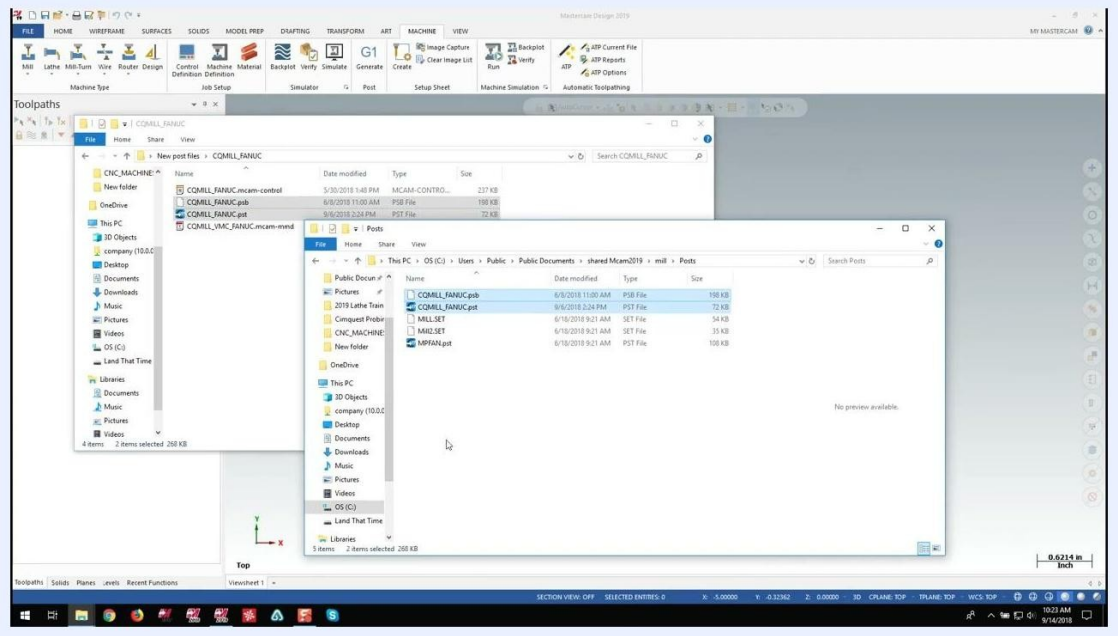
Mastercam 2019 with the MACHINE tab selected and the ribbon showing machine-related options

Step 7: Open the machine list manager and add your machine

After you install the custom post processor files, add the new machine to your active list.

1. In the **MACHINE** tab, click the **Mill** drop-down menu.
2. Select **Manage List**. The Machine Definition Menu Management dialog opens.
3. Review the two lists. The left side shows available machines. The right side shows machines currently available for selection in your workspace.
4. In the left list, locate your new machine (for example, CQMILL_VMC_FANUC.mcam-mmd).
5. Select the machine and click the **Add** button (the right arrow) to move it to the right-side list.

6. Click **OK** to confirm and close the dialog.

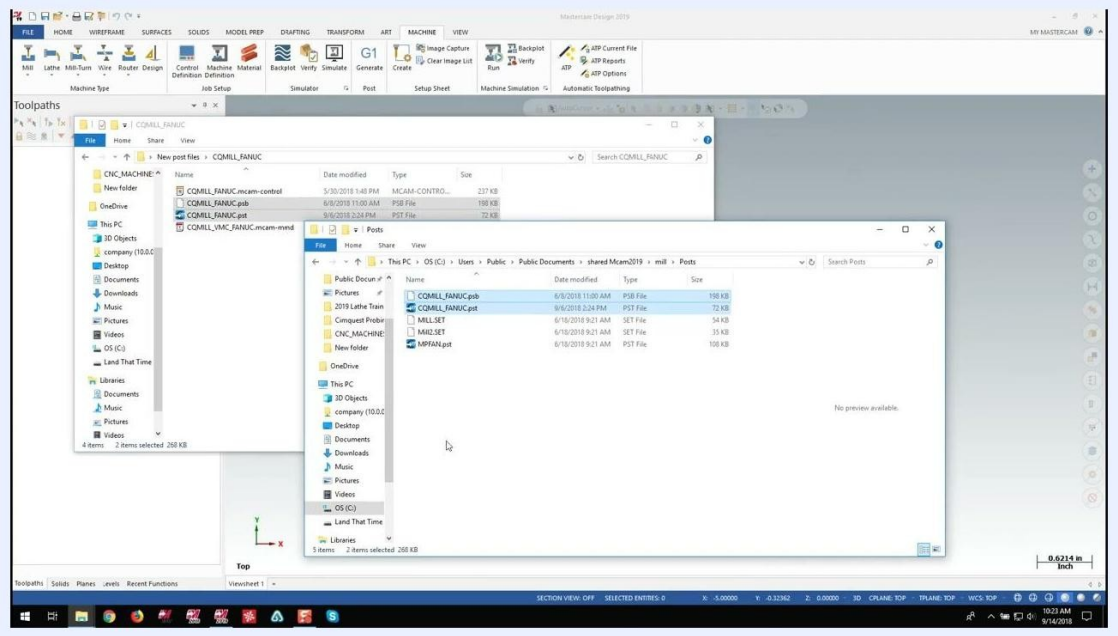


Machine Definition Menu Management dialog in Mastercam 2019 showing the machine directories and the OK button being clicked

Step 8: Load your machine and confirm it is ready

1. Go back to the **Mill** pull-down menu in the **MACHINE** tab.
2. Confirm that your new machine appears in the list of available machines.
3. Select your new machine to load it into your workspace.

Your new machine then appears in the Toolpaths Manager on the left side of the screen. For example, it can appear under "Machine Group - 1" as "Simquest VMC Mill Iso".



Mastercam 2019 workspace with the new machine loaded in the Toolpaths Manager and ready for programming

What's next

You have now installed the custom post processor in Mastercam 2019 and added its machine to your active list. You can begin programming with your new machine and post processor.