

# Installing a Post Processor

After installing a CAD/CAM program on your computer, you must tell the program that you are using a Techno CNC router so that it outputs the correct code. The way this works is through the usage of a post-processor.

To install our post processor, you must download the latest version from our website; [support.technocnc.com](http://support.technocnc.com)



From this site, navigate to the post processor page and select from the list your CAD/CAM program and machine type.

The screenshot shows a web browser window with the URL [support.technocnc.com/posts.htm](http://support.technocnc.com/posts.htm). The page header features the Techno CNC Systems logo, a "Back to Main" button, and a "Support" banner with a phone icon and contact information: "Call 631-648-7481 Mon-Fri 8:30 am EST to 5 pm EST".

The main content area is titled "Post Processors for Software Packages Techno CNC Routers" and includes the instruction: "Please click on the link applicable to your machine to download the zip file." Below this, there are several sections with yellow highlights:

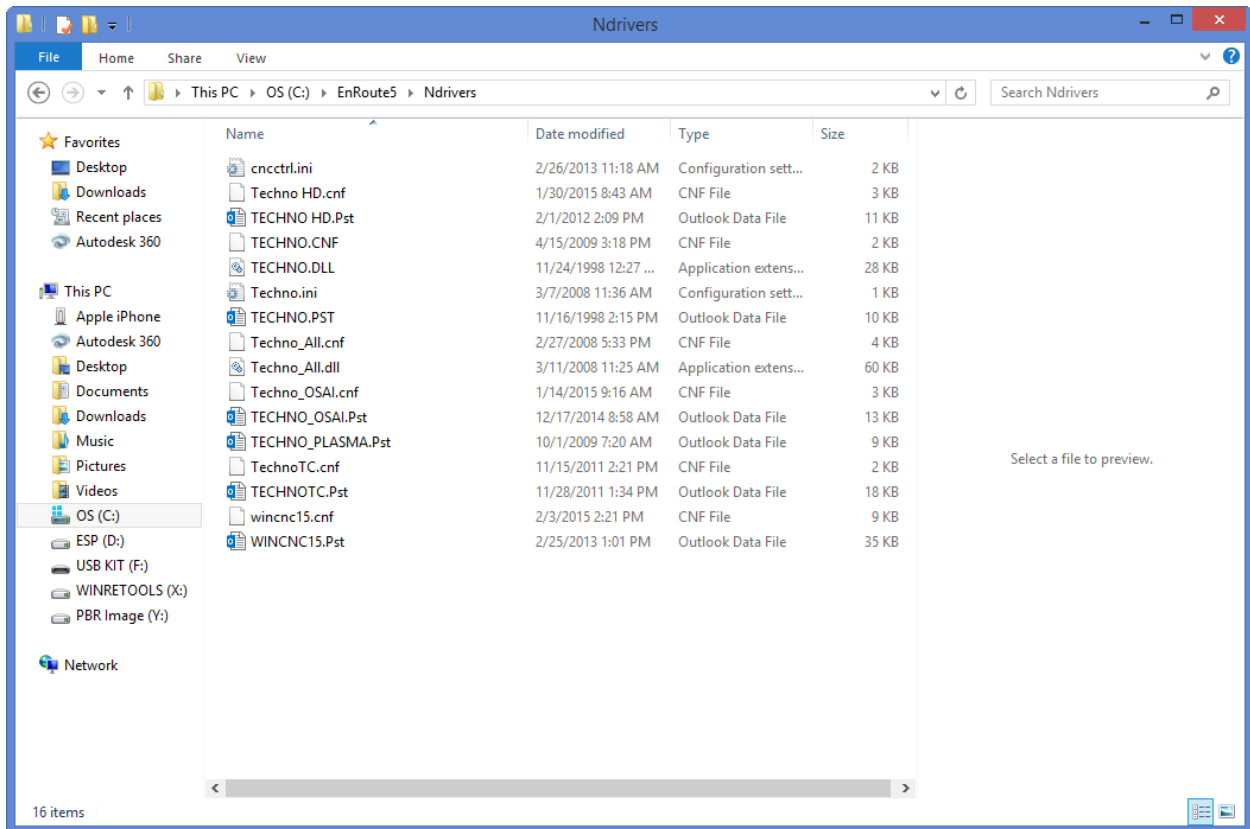
- Enroute**: Includes links for "HDS new 'Open' Osai controller" and "HDS '10 Series' Osai Controller".
- MasterCam Mill version X8**: Includes a link for "HD".
- MasterCam Router version X8**: Includes a link for "HD".
- Vetric for VCarve Pro or Aspire**: Includes a link for "HD" and a link for "HDS mm or HDS inch".
- VisualMill/RhinoCAM**: Includes a link for "HD" and a link for "HDS".

At the bottom of the page, there is a navigation menu with links: [Software Overview](#), [Control Interface](#), [ART Controller](#), [Vision System](#), [TechProbe](#), [TechScan](#), [Lathe](#), [Manuals](#), and [Contact Support](#). A "Back to Main" button is also present, and the footer contains the text: "Copyright © 2015 Techno CNC Systems, LLC. All rights reserved."

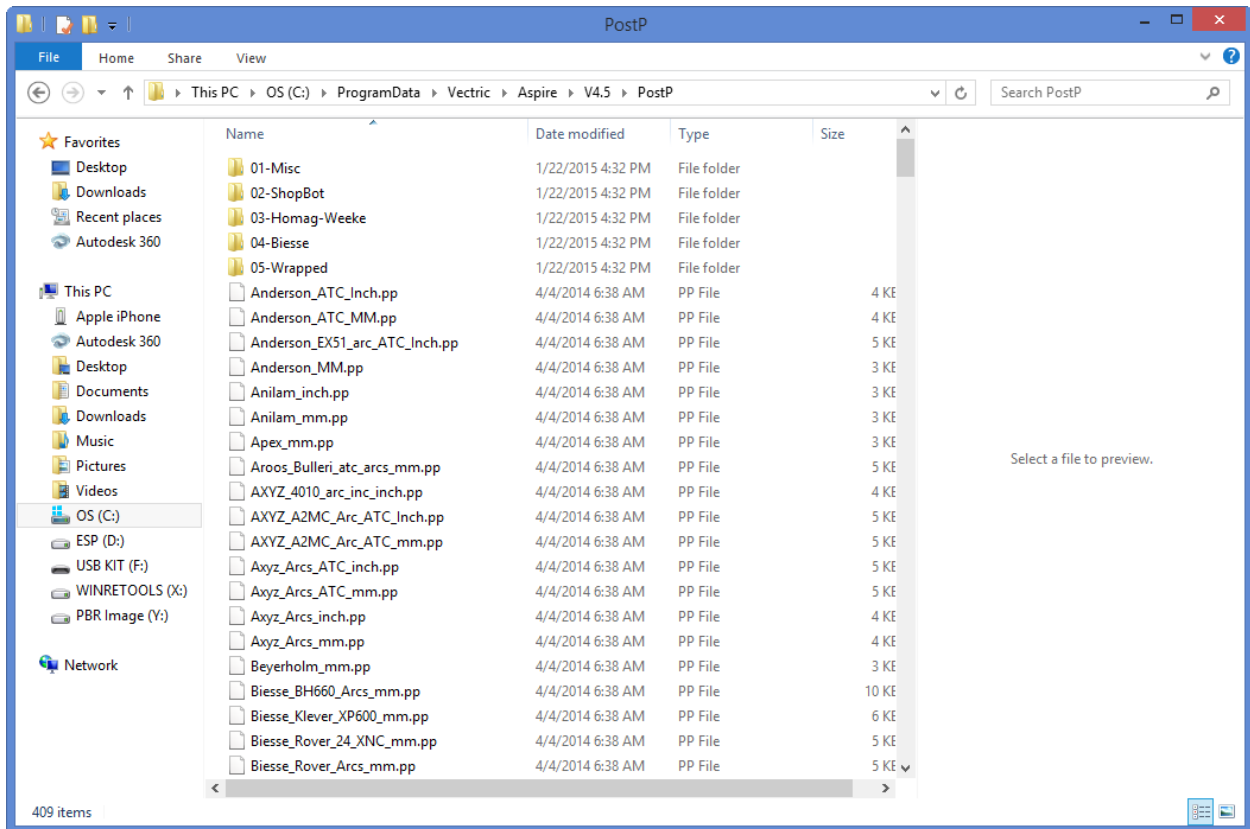
Note the save location of the file(s) you have downloaded because we will be moving these files to their correct locations based on what program you are using.

Please copy the post-processor files to the following locations;

Enroute:



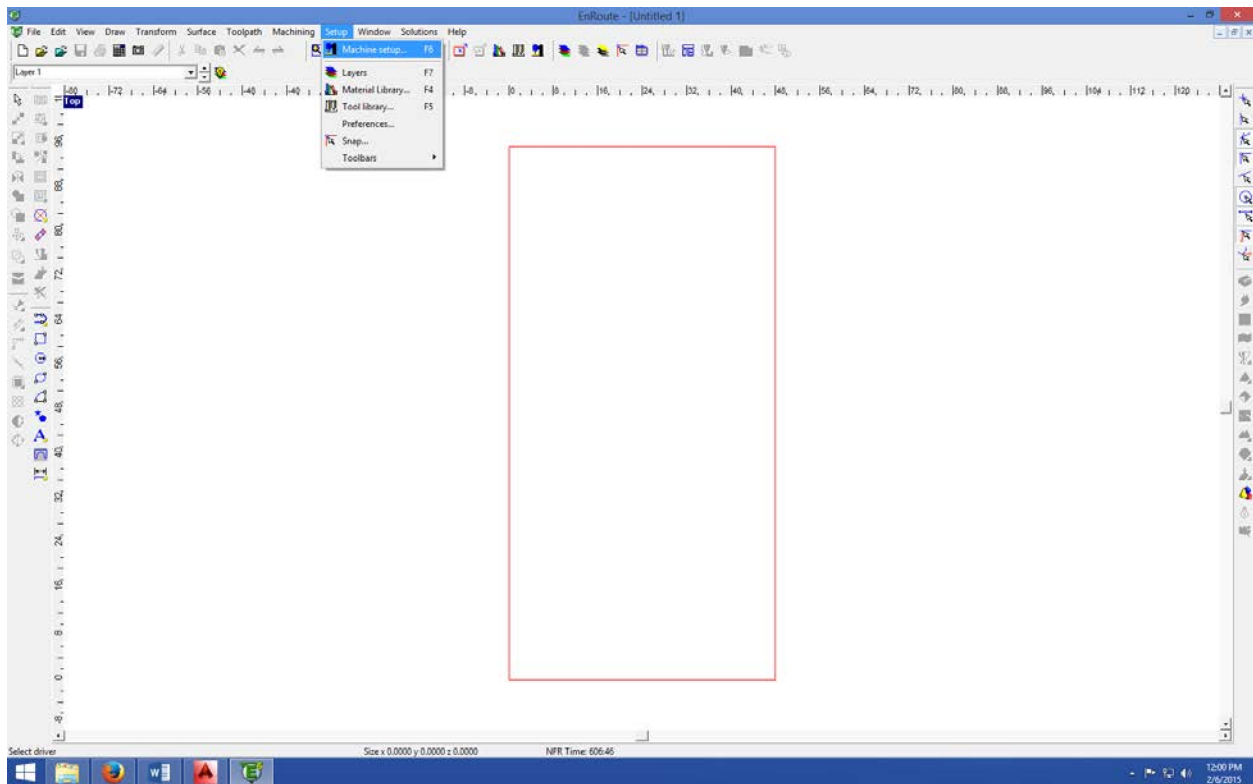
Vetric:



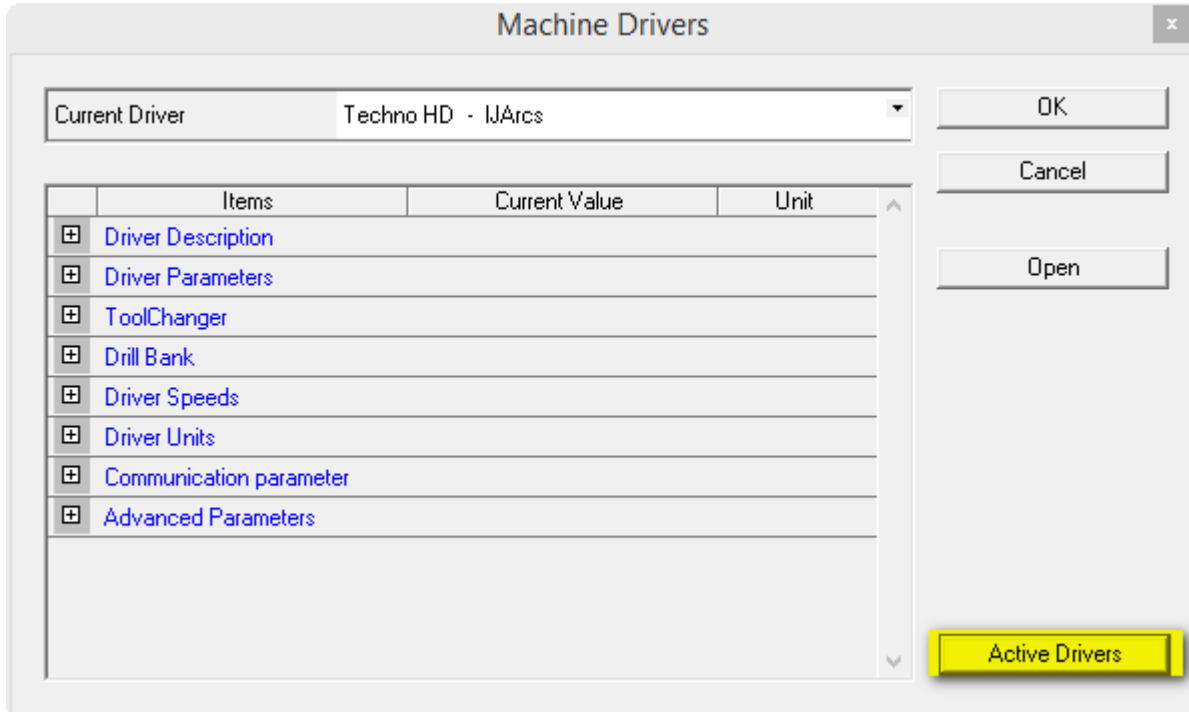
To use these post-processors, you must notify your CAD CAM program to select them by default:

Enroute:

Press F6 or Go to setup>Machine Setup



On the new pop-up screen, select “Active Drivers”



On this screen, please select the Techno CNC Post appropriate for your machine,  
“Techno\_Osai – Rarcs” for the HDS Machine

“Techno HD – IJArcs” for the HD Machine

And then select ok.

Vetric:

When you are done creating a toolpath and it is time to output the G-code file, after hitting “save toolpath”, you will select the appropriate Techno drivers from the pull-down list.

