CAM-POST Foundation

2 & 3 Axis Post-Processor Development

03 - Creating a Milling Post

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video we will demonstrate the creation and validation of a custom post-processor for a "***Haas VF-1***" 3-axes mill.

To configure the post, we will launch the *CAM-POST FOUNDATION* ***Quest*** module from the shortcut on the desktop.

…Press "*OK*" to remove the splash screen.

In the left window pane, we see the default database which initialized during the software installation. As you can see, no post-processors are yet stored in this database. Once we start configuring our new post-processor, the contents of the database will change.

Let's begin by clicking the "***New Post Wizard***" button on the top bar.

We are configuring a post for a "***Haas VF-1***" machine. Therefore we will name our post-processor ***HAAS_VF1***… We won't require an ID number, so we'll set the ***Post ID*** to "***Not Available***"… For traceability purposes, you can enter your name in the field if you want…

The ***machine type*** is "***Mill***" (this was the default) and we will set the ***post units system*** as "***Millimeters***"…

CAM-POST FOUNDATION comes with a built-in library of basic post-processors. These are templates you can use as a starting point when building your post… For our machine, we will use the "***HAAS Automation***"… milling template called "***Haas***". This will take care of the control-specific options.

For the ***machine defaults***, we will choose the standard "***VMC 3-axis***" template… A diagram explains the kinematic mapping of the axes.

Next, we enter the ***Machine Options*** dialog, where several tabs are available to configure options that are either machine-specific or reflect the personal preferences of the programmer… By hovering the mouse on top of the various fields, you can pop-up some explanations related to each option… G-codes used for rapid and feed moves… G-codes for arcs… Arc center point definition… and so on.

The ***Sequence Numbers*** tab allows you to specify the way you want the N-blocks to be numbered in the output programs. This is a typical example of a personal preference setting. Let's enter 5 as the "***initial block number***" …and 5 as an **increment**… Also, we will want block numbering to be used "***Always***".

The ***COOLANT*** tab lets you control the M-codes for each coolant type, and also when should the coolant codes be output by the post. We will select the option "***Plunge***" (the default), so the post will defer the M8 code until the first plunge move along the tool axis.

The ***Feedrate*** tab has two subsections: ***units-per-minute*** mode (G94), where the maximum feedrate on this machine is 16510 mm/min, and ***units-per-revolution*** mode (G95).
The ***Tape*** tab allows you to define the format of your tape output: whether you need a percent sign at the beginning and the end, whether you prefer to use spaces between registers, whether the program can contain lowercase characters… You can decide to output ***PPRINT statements*** as comments or as messages, and to define the start and end of message text (open and closed parentheses).

In the ***Spindle*** tab, we will enter the ***minimum*** and ***maximum RPM*** available on this machine: 10 and respectively 8100 RPM…

In the ***Tool change*** tab, we will set to 20 the ***maximum number of tools*** that are available on this machine…

Finally, we can take a look at the ***Registers*** tab, where all the letter addresses used by the post are listed. We can see a summary of the specific designation of each register on the last column to the right…

Next, we are entering the ***Post customization*** dialog, where *some* customization was already pre-defined in the Haas template file. There are four main areas where you can apply customization, if required: at the ***Start of the program***, ***Before a toolchange***, ***After the toolchange***, and at the ***End of the program***.

And with this, we've completed the first configuration of the post. Prior to exiting the wizard, we have an option to ***Generate*** the post. The Generate command compiles the post into the database and also validates there are no conflicts in the post settings. We'll leave this option checked, and click "***Complete***".

Back to the *CAM-POST FOUNDATION Questionnaire*, let's take a look in the ***Database*** tab on the left pane… Notice that while before the database was empty, it is now showing the post-processor HAAS_VF1, with its first version, HAAS_VF1.200;1. A new tab, with the name of the current post, allows us to make further edits if required. The name of the post also includes its version number (1).

Now, let us test the post to see what the output looks like. Click the "***Start***" button on the top menu bar. In the launch panel, click next to the CL file to select a test file produced by your CAM system. In this case, we will use a Mastercam NCI file named "3-axis_milling"…

The launch panel allows you to enter the ***Program ID***… Let's enter 1234.

Still on the launch panel, you can click the ***CAM*** button to specify the CAM system that generated your Cutter Location file. You can use "***Automatic***" to let *CAM-POST FOUNDATION* recognize the appropriate CAM interface to use… Or you can choose the interface by pulling-down the list… We'll use the "***Automatic***" option.

Click "***OK***" to exit the CAM selection, then click "***OK***" to launch the post-processor. Note that we're running the post-processor currently loaded in memory – our HAAS_VF1 – which is still "work-in-progress".

The ***Gener*** window is launched. On the left pane, we can see the input which the post "translated" from the Mastercam NCI file. On the right, we see the output codes that are being produced by the post… Notice the output of a toolchange… And the way the first rapid approach is broken down in two moves: XY, then Z with the length offset… We can click in any of these partitions to go one-step-at-a-time.

Also notice the format of circular interpolation codes: G2 is using the radius method…

Let us just run the program 'till the end by clicking the "***Play***" button ("*Process*") on the top menu bar... We can pause at any time… and resume… The level of completion is shown in the lower-right corner.

Let's scroll up to the top. It would be nice to have a tooling list at the beginning of the program…

No problem; without closing Gener, we can return to the Questionnaire, where the post is still in memory, and select the ***Machine Startup Macro*** under the ***Post Customization*** section… Scrolling down on the list of built-in actions, we'll find a pre-defined customization item labeled "***Print Tooling Summary***". Select and push it to the right by clicking the "***Add***" button… Then press "***OK***"…

Let us return to Gener and rewind the process, then run the post again. Since the post is in memory, the latest changes will apply… Notice the list of tools which is extracted automatically by scanning the NCI file…

Let's also change the circle definition: instead of radius, we will rather use the IJ offsets. Again, we will leave Gener open and return to the Questionnaire. In the "***Control Description***" section on the left, we will select "***Circular and Helical Interpolation***". We will disable the Radius method altogether…

Back to Gener, we will rewind and start again… Notice that now the circles are output using I and J…

We'll run the program to the end… Then exit Gener.

Back in the Questionnaire, it's about time we ***Generate*** the post again with the new changes… We'll click a button on the top menu bar… The name, again, is HAAS_VF1.

As a result, the database will now contain two versions of the post, numbered 1 and 2. Each time the post is generated, a new revision will be added automatically. After a while, you may only want to keep the last revision, discarding older ones. Right-click the node name of the post and select "***Purge***"… You have an option to rename the last revision back to 1, or leave it as it is. We'll leave it as revision 2, by pressing "***No***".

Our post is finished and ready for production.

This concludes our video tutorial on creating a 3-axes milling post in *CAM-POST FOUNDATION*.
Thank you for watching.

04 - Creating a Turning Post

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video we will demonstrate the creation and validation of a custom post-processor for an "***OKUMA LB-35***" 2-axes lathe.

To configure the post, we will launch the *CAM-POST FOUNDATION* ***Quest*** module from the shortcut on the desktop.

…Press "***OK***" to remove the splash screen.

In the left window pane, we see the default database, in which we have already created the "***HAAS_VF1***" milling post-processor. The new turning post will be added to the same database.

Let's begin by clicking the "***New Post Wizard***" button on the top bar.

We are configuring a post for an "***OKUMA LB-35-2***" turning center. Therefore we will name our post-processor ***OKUMA_LB35***… We won't require an ID number, so we'll set the ***Post ID*** to "***Not Available***"… For traceability purposes, you can enter your name in the field if you want…

The ***machine type*** is "***Lathe***", and we will set the ***post units system*** as "***Millimeters***"…

As previously seen, *CAM-POST FOUNDATION* comes with a built-in library of basic post-processors. These are templates you can use as a starting point when building your posts… For our machine, we will use the "***OKUMA***"… "***OSP Series L***" template. Again, this will take care of all the control-specific options.

For the ***machine defaults***, we will choose the "***LATHE 2H***" template, for a 2-axes horizontal lathe… A diagram explains the kinematics.

Next, we enter the ***Machine Options*** dialog, where several tabs are available to configure options that are either machine-specific or reflect the personal preferences of the programmer… Again, by simply hovering the mouse on top of various fields, you can pop-up some explanations…

Let's go to the ***Sequence Numbers*** tab to enter some personal preference settings. We'll enter 5 as the "***initial block number***" …and 5 as an ***increment***.

In the ***COOLANT*** tab, we will set how do we want the coolant to be output. Answer "***Yes***" to the question "***Output coolant with next motion***", so that the M8 code will show on the same block as the first move.

In the ***Feedrate*** tab, subsection ***units-per-minute*** (G94), we'll set the maximum feedrate to 15000 mm/min. No changes in the ***units-per-revolution*** section (G95).

In the ***Tape*** tab, we will verify some options related to the output format: messages will begin with an open parenthesis and end with a closed one, the program will use spaces between registers, it will only contain uppercase characters… and so on.

In the ***Spindle*** tab, we will set the maximum RPM to 3200 RPM… We will also verify the settings in the ***Constant Surface Speed*** subsection: G96 for SMM, G50 for MAXRPM…

Next, we will go to the ***Turret*** tab to set the ***total number of tools*** to 12… We'll leave the ***number of tool offsets*** to 9999.

Finally, we'll take a quick look at the ***Registers*** tab, to see a list of all the letter addresses used by the post along with their specific designation…

The ***Post customization*** dialog will, once again, contain *some* customization which was pre-defined in the OSP template file. The four main areas where you can apply customization are: at the ***Start of the program***, ***Before a toolchange***, ***After the toolchange***, and at the ***End of the program***.

With this, the post configuration is completed. Prior to exiting the wizard, we will select the option to ***Generate*** the post. This option compiles the post into the database and validates there are no conflicts in the settings… And we'll click "***Complete***".

Back to the *CAM-POST FOUNDATION* *Questionnaire*, let's take a look in the ***Database*** tab on the left pane… Notice that now there are two posts: the pre-existent HAAS_VF1 milling post-processor, and the first version of the new turning post, OKUMA_LB35.200;1.

Now, it is time to test the post. Click the ***Start*** button on the top menu bar. In the launch panel, click next to the CL file to select a turning test file produced by the CAM system. In this case, we will use a Mastercam NCI file named "2-axis_turning"…

We'll click the ***CAM*** button to verify the CAM interface is set to "***Automatic***". This will tell *CAM-POST FOUNDATION* to automatically recognize the CAM interface…

Click "***OK***" to exit the CAM selection, and click "***OK***" to launch the post-processor. Note that we're running the post-processor currently loaded in memory.

The ***Gener*** window is launched. On the left pane, we can see the input which the post "translated" from the Mastercam NCI file. On the right, we see the output codes that are being produced by the post… Notice the output of a toolchange… We can click in any of these partitions to go one-step-at-a-time.

Let's click the ***Play*** button to run the program completely... then scroll up to the top. It would be nice to have a tooling list at the beginning of the program…

No problem; without closing Gener, we return to the Questionnaire, select the ***Machine Startup Macro***, scroll down through the list of built-in actions until we find a pre-defined customization item labeled "***Print Tooling Summary***". Select and push it to the right by clicking the "***Add***" button… Then press "***OK***"…

We'll return to Gener, rewind the process, then run the post again. Since the post is in memory, the latest changes will apply… Notice the list of tools which is extracted automatically by scanning the NCI file…

We'll run the program to the end… Then exit Gener.

Back to the Questionnaire, let's ***Generate*** the post with the new changes… The name, again, is "***OKUMA_LB35***".

The database will now show <u>two</u> versions of the Okuma post, numbered 1 and 2. We can discard the older one by right-clicking the node name of the post and selecting "***Purge***"… There is an option to rename the last revision back to 1, or leave it as it is. We'll leave it as is, so we'll press "***No***". Our post is now finished and ready for production.

This concludes our video tutorial on creating a 2-axes turning post in *CAM-POST FOUNDATION*.
Thank you for watching.

05 - Setup for CATIA

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video we will demonstrate the integration of *CAM-POST FOUNDATION* to CATIA V5 or V6.

In order to make CATIA use *CAM-POST FOUNDATION* to generate NC code, you must run a "***Setup***" utility.

To run this utility, press the Windows "*Start*" button, select "All programs",
…"CAM-POST FOUNDATION V20",
…"Setup",
…"Kit",
…and select "CATIA".

The ***ICAM Integration to CATIA*** dialog appears. This dialogue has two tabs that are available: "***General***" and "***GENER***".

The "***General***" tab contains two sections: "***ICAM Software***" and "***Database***". The first is used to specify which version of CAM-POST should be used. Since the system automatically detects the *CAM-POST FOUNDATION* installation, you can leave this section unchanged.

The second section allows you to select the default database to be used when running a *CAM-POST FOUNDATION* post-processor within CATIA. You can either use the default database initialized during the software installation, or use your own, custom database.

In this example, we will use the default ICAM database.

Now let us move on to the second tab. Click "***GENER***".

This tab allows you to select the interface mode to be used when running Gener within CATIA. You can choose to run it in the background, or with an interface.

By default, the ***Background*** option is selected. Uncheck the checkbox.

You can now select the type of interface you want to use. Either the full interface,
…a progress bar,
…or minimized.

Select the ***Progress*** option.

When done, press OK to exit the setup.

This concludes this video tutorial on integrating *CAM-POST FOUNDATION* to CATIA.
Thank you for watching.

06 - Setup for Mastercam (1)

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video we will demonstrate the integration of *CAM-POST FOUNDATION* to Mastercam X7.

In order to make Mastercam use *CAM-POST FOUNDATION* to generate NC code, you must run a "Setup" utility.

To run this utility, press the Windows "Start" button, select "All programs",
…"CAM-POST *FOUNDATION* V20",
…"Setup",
…"Kit",
…and select "Mastercam X+".

The ***ICAM Integration to Mastercam*** dialog appears. This dialogue has 3 tabs that are available: "***General***", "***Mastercam***" and "***GENER***".

The "***General***" tab contains two sections: "***ICAM Software***" and "***Database***". The first is used to specify which version of CAM-POST should be used. Since the system automatically detects the *CAM-POST FOUNDATION* installation, you can leave this section unchanged.

The second section allows you to select the default database to be used when running a *CAM-POST FOUNDATION* post-processor within Mastercam. You can either use the default database initialized during the software installation, or use your own, custom database.

In this example, we will use the default ICAM database.

Now let us move on to the second tab. Click "***Mastercam***".

The system automatically detects all Mastercam versions installed on your workstation. If more than one version is installed, it will select the newest one. If required, you can pull-down the list and select the desired Mastercam version. We'll select "***Mastercam X7***"…

The system also detects the Mastercam installation folder, the shared directory and the user directory. Just verify they are correct.

Next, we will generate custom PST files for all the posts that were configured so far. Press the ***Generate*** button to do so…

Another dialog pops up, ***Update PST Posts***. In here, you'll have a list of all the post-processors that were created in the database you selected at the previous step. In this case, we'll have the HAAS_VF1 and the OKUMA_LB35 lathe post. Press the ***Update*** button to generate PST files for these posts… A message is informing you that the post installation is completed…

Finally, go to the thrid tab, "***GENER***"…

This tab allows you to select the interface mode to be used when running Gener within Mastercam. You can choose to run it in the background, or with an interface.

By default, the ***Background*** option is selected. Uncheck the checkbox.

You can now select the type of interface you want to use. Either the full interface,
…a progress bar,
…or minimized.

Select the ***Progress*** option.

When done, press OK to exit the setup.

This concludes this video tutorial on integrating *CAM-POST FOUNDATION* to Mastercam.
Thank you for watching.

07 - Setup for Mastercam (2)

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video we will demonstrate the setup of the Mastercam Machine and Control files.

During the previous step, we have created a PST file for the ***HAAS_VF1*** custom post-processor built with CAM-POST *FOUNDATION*.

We will launch Mastercam and select ***Settings*** → ***Machine Definition Manager***.

We will open an existing machine definition to start with. Press the "***Open***" icon.

Depending on the machine kinematics, we should select an appropriate existing machine definition, such as a 3-axes one. However, in this case, we will select the "***MILL DEFAULT MILLIMETER.MMD-7***" configuration.

First of all, you should do a "***Save As***" to save the original machine under a different name, so it will not be overwritten by our custom definition. We will save it as "***HAAS_VF1_ICAM.MMD-7***".

Press the "***Edit the control definition…***" button.

In the new dialog, you should once again do a "***Save As***" to save the original control under a different name. We will save it as "***HAAS_VF1_ICAM.CONTROL-7***".

Optionally, you can enter the control manufacturer name (e.g. ***Haas Automation***) and a short description of the control.

Press the "***Post processors***" button.

In the new Post List Edit dialog, select the existing "***MPFAN.PST***" post and press "***Delete files…***".

Press "***Add files…***" and select the "***HAAS_VF1.PST***" file generated with the *Setup* utility.

Click ***OK*** to return to the *Control definition* dialog, then pull-down the list of post-processors and select the new PST file we just added.

In the *Control topics* pane at the left of the dialog, select ***Files*** and pick the option ***Post executable***.

Press the cabinet icon to the right of the field. In the file selection dialog, select the "***icamx7.dll***" file and press ***Open***.

Press the "***Save***" button on the top menu to save the control, then press OK to exit the *Control definition* dialog.

Back to the *Machine Definition Manager*, enter a machine description (e.g. ***Haas VF-1***) in the *Description* field.

Finally, press the "***Save***" button to save the machine, then press ***OK*** to exit the dialog.

Accept the prompt to replace the existing machine in the group. The new post is now shown in the group properties.

It is also ready to use…

This concludes our presentation of the setup of Mastercam Machine and Control files.
Thank you for watching.

08 - Setup for NX

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video we will demonstrate the integration of *CAM-POST FOUNDATION* to Siemens NX CAM.

In order to make NX use *CAM-POST FOUNDATION* to generate NC code, you must run a "***Setup***" utility.

To run this utility, press the Windows "Start" button, select "All programs",
…"CAM-POST FOUNDATION V20",
…"Setup",
…"Kit",
…and select "UGS NX".

The ***ICAM Integration to UGS NX*** dialog appears. This dialog has 3 tabs that are available: "***General***", "***UGS NX***" and "***GENER***".

The "***General***" tab contains two sections: "***ICAM Software***" and "***Database***". The first is used to specify which version of CAM-POST should be used. Since the system automatically detects the *CAM-POST FOUNDATION* installation, you can leave this section unchanged.

The second section allows you to select the default database to be used when running a *CAM-POST FOUNDATION* post-processor within NX CAM. You can either use the default database initialized during the software installation, or use your own, custom database.

In this example, we will use the default ICAM database.

Now let us move on to the second tab. Click "***UGS NX***".

The system automatically detects the newest NX installation on your workstation. It also detects the NX ***installation***, ***post-processor*** and ***tool path*** folders. Just verify they are correct.

Next, we will generate custom files for all the post-processors that were configured so far. Press the ***Install / Update Post-processor***" button to do so…

Another dialog pops up, ***Install Post-processor to NX***. In here, you'll have a list of all the post-processors that were created in the database you selected at the previous step. In this case, we'll have the HAAS_VF1 and the OKUMA_LB35 lathe post.

Ensure that the ***CLSF Format*** option is set to "***CLSF Advanced***". Pull-down the list and select this option.

Press the "***Install / Update***" button to generate all posts. A message will inform you, for each post, that the installation is completed. Press "***OK***" to continue.

Press "***Close***" to exit the dialog.

Finally, go to the thrid tab, "***GENER***"…

This tab allows you to select the interface mode to be used when running Gener within NX. You can choose to run it in the background, or with a visual interface.

By default, the "***Background***" option is selected. Uncheck the checkbox.

You can now select the type of interface you want to use. Either the full interface, a progress bar, or minimized.

Select the "***Progress***" option.

When done, press "***OK***" to exit the setup.

This concludes this video tutorial on integrating *CAM-POST FOUNDATION* to Siemens NX CAM. Thank you for watching.

09 - Running a Post from GENER Shortcut

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video, we will demonstrate the execution of an existing post-processor directly from the operating system.

In a previous video (Video #3), we have configured a 3-axes milling post for a ***Haas VF-1*** machine. Let us now run this post-processor on an existing Cutter Location Data file…

Click the ***GENER*** shortcut, which was created on the desktop during the *CAM-POST FOUNDATION* installation…

This opens up the *"**CAM-POST GENER**"* launch panel… Here, we can select the Cutter Location file to process… Let us select a Mastercam-generated ***NCI*** file... and press "***Open***".

We can also specify the *CAM-POST FOUNDATION* database to use… and select the post.

The CAM button allows us to specify the CAM system that generated the Cutter Location file… In this case, we will use the "***Automatic***" option, which tells *CAM-POST FOUNDATION* to automatically recognize the format of the CL file and use the appropriate CAM interface…

"***Exit***" the *CAM Interface* dialog…

We can select the "***Progress***" option, or launch the process with a full verbose interface… Let's use the *Progress* bar… and press ***OK***.

The GENER progress bar appears... At any time, we can select "***More Info***" to access the full GENER interface.

Let us rewind the process and run it once again…

The GENER interface has several different panels, in which we can click to execute just one step at a time…

…Here in the ***Input*** window, we see the CL records that the software has converted from the NCI file… We can see motion data like GOTO points or CIRCLES…

…In the ***Output*** window, we can step through the NC blocks that are currently generated by the post-processor…

…There is also a ***Console*** window, in which warnings and errors are displayed when found…

…We can resume the continuous process and pause it again as needed…

All of these windows are interconnected, and can be synchronized by right-clicking an item in a window and picking ***Synchronize***… The corresponding item in the other window is underlined…

We can select a line in the CL Data file… and synchronize it with the resulting NC block… We can then *step-in* to the next line… and the next… and trace the output one-step-at-a-time.

We can do this the opposite way, by picking a *block* in the NC output and synchronizing it with the CL command that generated it.

Let us now select the ***View Listing*** button, which allows us to verify the *Post Verification Listing* file. This file can be optionally output in HTML format, in which case we'll have some links that allow us to quickly access the beginning of each tool change…

…and verify, on every block, the current XYZ position.

We can also go directly to the summaries section… we'll have a ***tooling summary***… a ***travel summary***… the ***diagnostics***… As you can see no diagnostics were found during this process…

Finally, let us take a look at the actual *NC program* that was generated. Let's click ***View Tape File***… and scroll down in the editor to quickly verify the file…

This concludes our presentation on running a *CAM-POST FOUNDATION* post directly from the shortcut on the operating system.
Thank you for watching.

10 - Running a Post from CATIA

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video, we will demonstrate the execution of an existing post-processor from CATIA.

At this time, the *CAM-POST FOUNDATION setup for CATIA* should have already been completed. In case you have not yet performed this step, please refer to the video #5 in the series.

First, we must insure that CATIA is using *CAM-POST FOUNDATION* as a post-processor OIM solution. In the CATIA top menu bar, click ***Tools*** → ***Options***. In the CATIA *Options* dialog, select ***Machining*** on the options list and activate the ***Output*** tab. Select the ***ICAM*** radio button in the Post-processor and Controller Emulator Folder.

Next, let's edit the Part Operation properties in the CAT Process. Press the ***Machine*** button… and activate ***Numerical Control***.

As a result of the *setup* procedure previously executed, the "***HAAS VF1 200***" post was included on the list of *available post-processors*. Our example here is set to use *this* particular post-processor.

Insure that the "***ICAM _MM_ V20. PPTable***" is selected as a *post-processor words table*… and that the *NC Data Type* is set to ***ISO***.

Click ***OK*** to exit the *Machine Editor*, then click ***OK*** to exit the *Part Operation* dialog.

Now, let us *post-process* the operation… Right-click the manufacturing program (in this case it's called ***3-Axes***), and select "***Generate NC Code Interactively***" from the *3-Axes object* pull-down list.

In the dialog, verify the *selection*… the NC Data Type… and the name of the output file. When satisfied, press "***Execute"***.

The *"**CAM-POST GENER***" progress bar appears... We can see the completion as the process goes on.

At the top of the dialog, we can also see if any diagnostics were found during the process… Warnings, errors, fatal errors… As you can see, this program was error-free.

However, if any errors *would* have been reported, clicking ***View Listing*** would allow us to verify the Post Verification Listing file. This file can be optionally output in HTML format, in which case, we'll have some links that will allow us to quickly acess the beginning of each tool change…

…and verify, on every block, the current XYZ position.

We can also go directly to the ***summaries*** section… we'll have a ***tooling summary***… a ***travel summary***… the ***diagnostics***… and the program length.

When done reviewing, we can close the browser… press "***Exit"*** to end the *GENER* process… accept the notification in CATIA, and finally "***Close"*** the dialog.

This concludes our presentation on running a *CAM-POST FOUNDATION* post from CATIA.
Thank you for watching.

11 - Running a Post from Mastercam

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video, we will demonstrate the execution of an existing post-processor from Mastercam.

At this time, the *CAM-POST FOUNDATION setup for Mastercam* should have already been completed. In case you have not yet performed this step, please refer to videos #6 and #7 in the series.

As a result of the *setup* procedure, a "***HAAS VF1 ICAM MMD7***" machine configuration was created. Our example here is using *this* particular machine configuration, along with the post-processor that was associated with it.

Therefore, when we click the "G1" icon to post-process the selected operations, "***HAAS VF1 . PST***" is shown as the *active post*.

Let us accept the "***NC File***" selection and press ***OK***... As we chose to be prompted for a file name, we'll have to accept the warning in case the file already exists.

The *CAM-POST "**GENER**"* progress bar appears... We can see the completion as the process goes on (here, it only took a few seconds).

At the top of the dialog, we can see if any diagnostics were found during the process… Warnings, errors, fatal errors… This program was completely error-free, so let us just click "***Exit***".

Mastercam will now launch the associated program editor (in this case, it's *Code Expert*). We can take a look at the program we just generated… Notice the tooling summary that was configured in the post we created at the previous step.

When done, we can close the program editor.

This concludes our presentation on running a *CAM-POST FOUNDATION* post from Mastercam.
Thank you for watching.

12 - Running a Post from NX

Welcome to this series of *CAM-POST FOUNDATION* video tutorials. In this video, we will demonstrate the execution of an existing post-processor from Siemens NX CAM.

At this time, the *CAM-POST FOUNDATION setup for UGS NX* should have already been completed. In case you have not yet performed this step, please refer to the video #8 in the series.

First, we must insure that the files generated during the *Setup* procedure are now available within NX as post-processor files. Select a program in the ***Operation Navigator*** and click the ***Post-Process*** button on the top menu bar. In the *Post-Process* dialog that pops up, we can see that the "***OKUMA_LB35***" and "***HAAS _VF1***" posts are now showing at the top of the list of available post-processors.

Let's select the "***HAAS _VF1***" option.

The *"**CAM-POST GENER***" progress bar appears... We can see the completion as the process goes on.

At the top of the dialog, we can also see if any diagnostics were found during the process… Warnings, errors, fatal errors… As you can see, this program was completely error-free.

Let's press "***Exit"*** to end the *GENER* process...

NX now opens the resulting NC program in a text editor, so we can review the output… Everything seems correct…

When done reviewing, we can close the editor…

This concludes our presentation on running a *CAM-POST FOUNDATION* post from Siemens NX.
Thank you for watching.