What's New in CAMWorks 2026

October 2025

Table of Contents

(Click a link below or use the bookmarks on the left.)

What's New in CAMWorks 2026 SP0

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2026 - SOLIDWORKS 2025 - CAMWorks Solids 2026 - CAMWorks Solids 2025 - **3D**EXPERIENCE SOLIDWORKS R2026x - **3D**EXPERIENCE SOLIDWORKS R2025x
Operating System:	64-bit version of: - Windows 11 - Windows 10 [*Home Editions are not supported] **Note:** CAMWorks 2026 is supported only on 64-bit Operating systems.

Resolved CPRs Document

Purpose of Document:	The Resolved CPR (*CAMWorks Problem Report*) document has been updated to report the software errors that have been resolved in the current Service Pack (SP0).
Path to Document:	To view the document, select: *C:\Program Files\CAMWorks2026x64\CAMWorks_VC143\Lang\English\CW2026BuildInfo.pdf*

Discontinuation of MS Access Based TechDB

The ***SQLite*** based TechDB is the default TechDB format linked to the *CAMWorks* application at the time of installation/upgradation. In the CAMWorks versions prior to *CAMWorks 2026*, the other database formats supported were the ***SQL Server*** based TechDB and ***MS Access*** based TechDB formats.

From *CAMWorks 2026* version onwards, the only database format other than the default SQLite based TechDB format will be the SQL Server based TechDB format. The *MS Access* based TechDB can no longer be linked to *CAMWorks 2026* and later versions.

Once you upgrade your *CAMWorks i*nstallation to *2026* or any later version, only the *SQLite* based TechDB or *SQL Server* based TechDB can be assigned as the default TechDB linked to the application. If the default TechDB previously linked to your CAMWorks installation was an *MS Access* based TechDB containing customized TechDB data, then you will need to import this data into the *SQLite*/*SQL Server* based TechDB once you upgrade to a newer version of the *CAMWorks* application.

Support for Swiss Lathes in CAMWorks

Support for Swiss Turn Machines (Multi-Function Lathes) in CAMWorks

Swiss Turn Machine: A Modern Manufacturing Marvel

In the world of modern manufacturing, the demand for compact, precise, and intricate parts continues to rise — especially in industries like medical devices, aerospace, electronics, and watchmaking. To meet these challenges, engineers and machinists have long turned to a special kind of turning center: the **Swiss-Type Lathe**, often simply known as the **Swiss-Turn**.

Swiss Turn: A Machine Born for Precision

The story of the Swiss-Turn begins in the Swiss watchmaking industry, where the need for finely detailed components led to the development of a revolutionary approach to turning. Unlike traditional lathes, where the workpiece (stock) stays fixed and tools move to cut it, the Swiss-Turn machine does the opposite: **the bar stock itself moves longitudinally through a guide bushing**. This setup brings the cutting action close to the support point, drastically reducing deflection, especially important for long, slender parts.

This design unlocked a world of possibility: machinists could now produce extremely detailed components with higher accuracy, better surface finish, and improved efficiency. Over time, Swiss-Turn machines evolved to support **multi-axis** operations, **live tooling**, and **sub-spindle machining**, making them incredibly versatile and capable of complete part machining in a single cycle.

Advantages of Swiss Turn Machines

- **Precision:** Capable of maintaining tighter tolerances
- **Speed:** Multiple operations running simultaneously across turrets and spindles
- **Efficiency:** Less need for secondary operations or setups
- **Durability:** Lower tool wear due to reduced vibration and stable machining
- **Complexity Made Simple:** Ideal for parts with intricate geometries and tight specs
- **Lesser Monitoring:** Machines can operate and produce parts without any human supervision

CAMWorks Workflow for Machining with Swiss Turn Machines : Tailored for Success

Let's follow the journey of a part as it moves through the CAMWorks Swiss-Turn process with the example given below:

Step 1: The Design

Consider that an engineer designs a complex medical implant. The part is slender, with multiple grooves, drilled holes, and fine details—perfect for machining using a Swiss Turn machine.

Step 2: Machine Setup

Inside CAMWorks, the user selects a pre-configured Swiss Turn machine with Main and Sub spindles and tool posts. The stock and work holding setups are defined.

Step 3: Toolpath Generation

Using its Feature-Based Machining functionality, CAMWorks identifies machining features—such as outer and inner turning diameter, grooves, cross holes, etc., and automatically generates the appropriate toolpaths. Each operation is assigned to the correct turret or spindle, and machining strategies are optimized.

Step 4: Synchronization Magic

CAMWorks' Sync Manager functionality will let you coordinate every move between the spindles and turrets on a graphical timeline. Need to rough-turn on the Main spindle while drilling on the sub-spindle? Just drag and drop an operation for synchronization with another required operation. Transferring the part to the sub-spindle is just a few clicks away.

Step 5: See It Before You Cut It
The complete operation sequence is run through the machine simulation. You can view and verify while tools move in synchronization, and the part takes shape. You can check for collisions, optimize the cycle time, and make real-time adjustments.

Step 6: From Code to Cutting
Finally, CAMWorks will post-process the toolpaths into ready-to-run NC code, customized for Swiss-Turn machines. The part program will then be transferred to the shop floor for precision production.

The fast cycle times and flexibility of Swiss Turn machines make them the standard in high precision manufacturing applications.
Overall, you can trust CAMWorks to deliver on efficiency, precision, correctness, and control when it comes to Swiss Turn Machine Programming!

The following pages of this document cover the various functionalities and user interfaces introduced/modified in CAMWorks to support the machining of parts using ***Swiss Turn*** machines.

Machine Components & Definition, Tool Cribs, and Setup

Forms to create, edit and save Swiss Lathe/Multi-Function Lathe Machine Definitions within TechDB

Purpose:

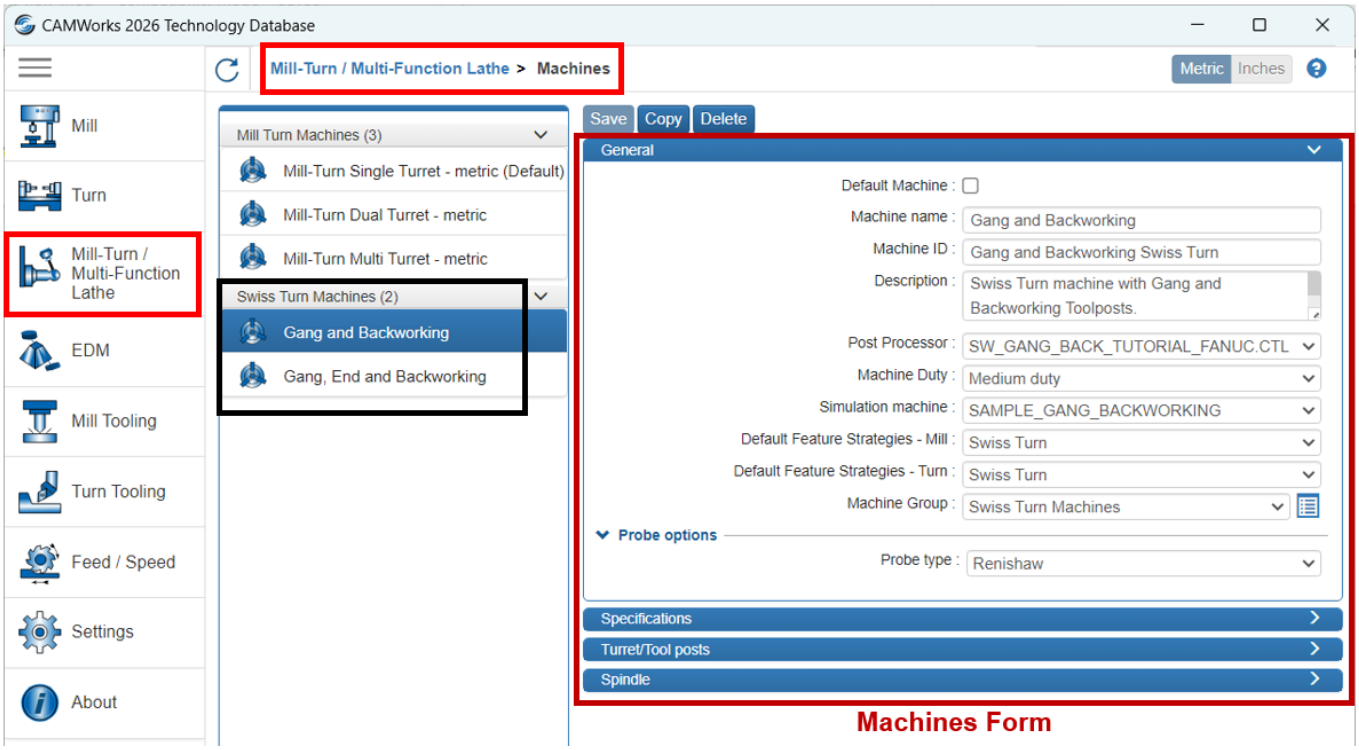
Provision to view, create, edit, and save Swiss Lathe/Multi-Function Lathe Machine definitions within the Technology Database

Implementation:

From *CAMWorks 2026* version onwards, within the Technology Database, the ***Mill-Turn*** menu on the left has been renamed to ***Mill-Turn/Multi-Function Lathe*** menu. The Mill-Turn machines and Swiss Lathes can be viewed, defined, edited, and saved within the ***Machines*** form available under this menu.

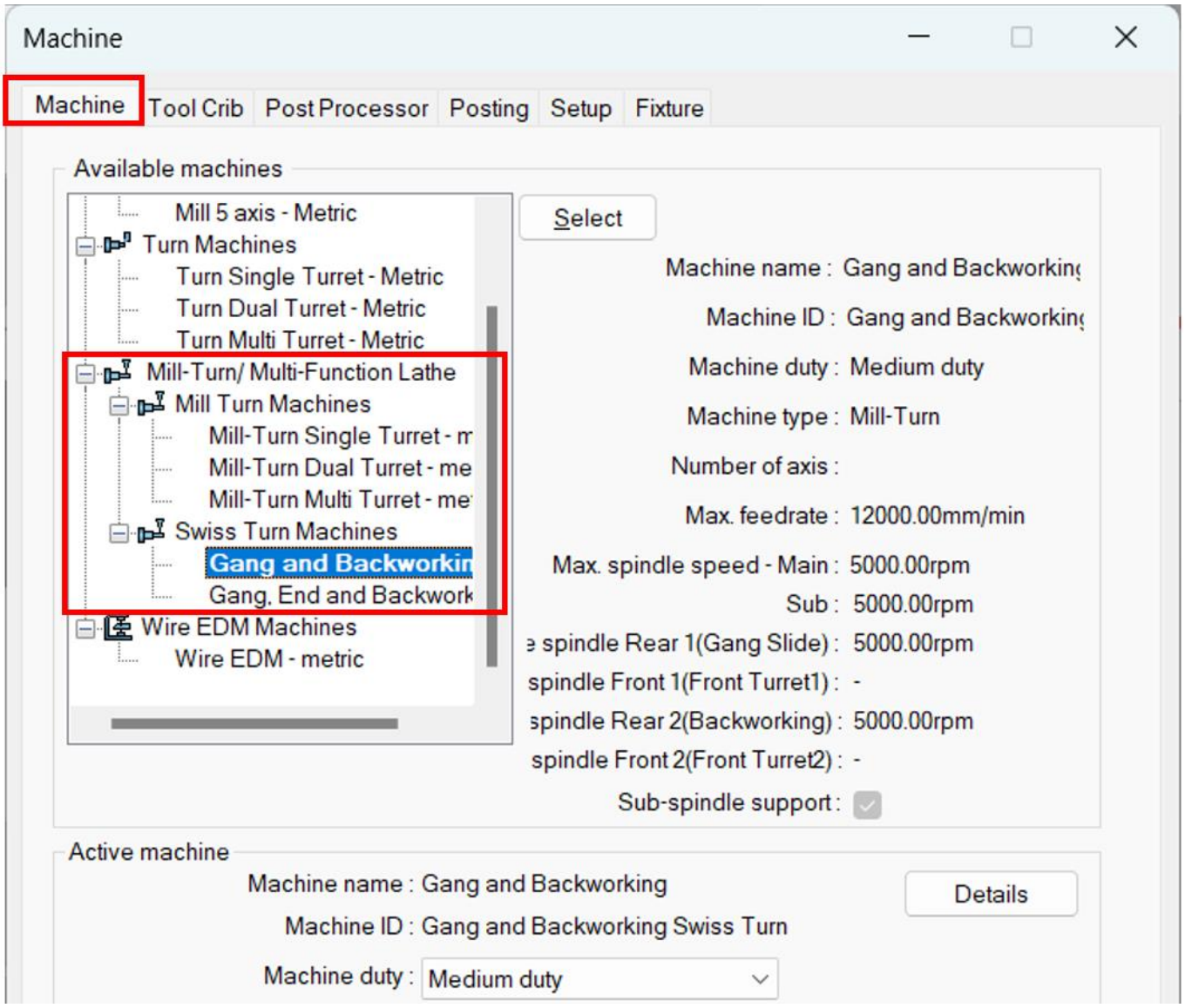
All sample Mill-Turn/Swiss Lathe machines shipped along with CAMWorks are listed within the ***Machines*** form under this menu. To create the machine definition corresponding to the Swiss Lathes/Multi-Function Lathes used at your machining facility within the TechDB, follow the steps bellow:

1. Within the TechDB, click on the *Mill-Turn/Multi-Function Lathe* menu.
2. Under *the Machines* sub-menu with the displayed interface, select any *Machine Definition* and click on the ***Copy*** button at the top of the interface.
3. The newly created Machine Definition will be listed under the *Machines* sub-menu and will be auto selected.
4. Use the ***Machines*** form to edit the various parameters associated with the machine definition and save the changes by clicking on the ***Save*** button. This action will save the machine definition.



Mill-Turn/Multi-Function Lathe>Machines Form of TechDB to view, create, edit, and save Swiss Lathe and Multi-Function Lathe Machine Definitions

When programming a part or assembly using the CAMWorks application, this newly added machine will become available, along with other saved machines in the TechDB, for machine selection under the ***Machine*** tab of the ***Machine*** dialog box.



Mill-Turn/Multi-Function Lathe Machines saved within TechDB available for Selection in CAMWorks under Machine Tab of Machine Dialog Box

Machine Groups Option to sort and classify Mill-Turn/Multi-Function Lathe Machines

Purpose:

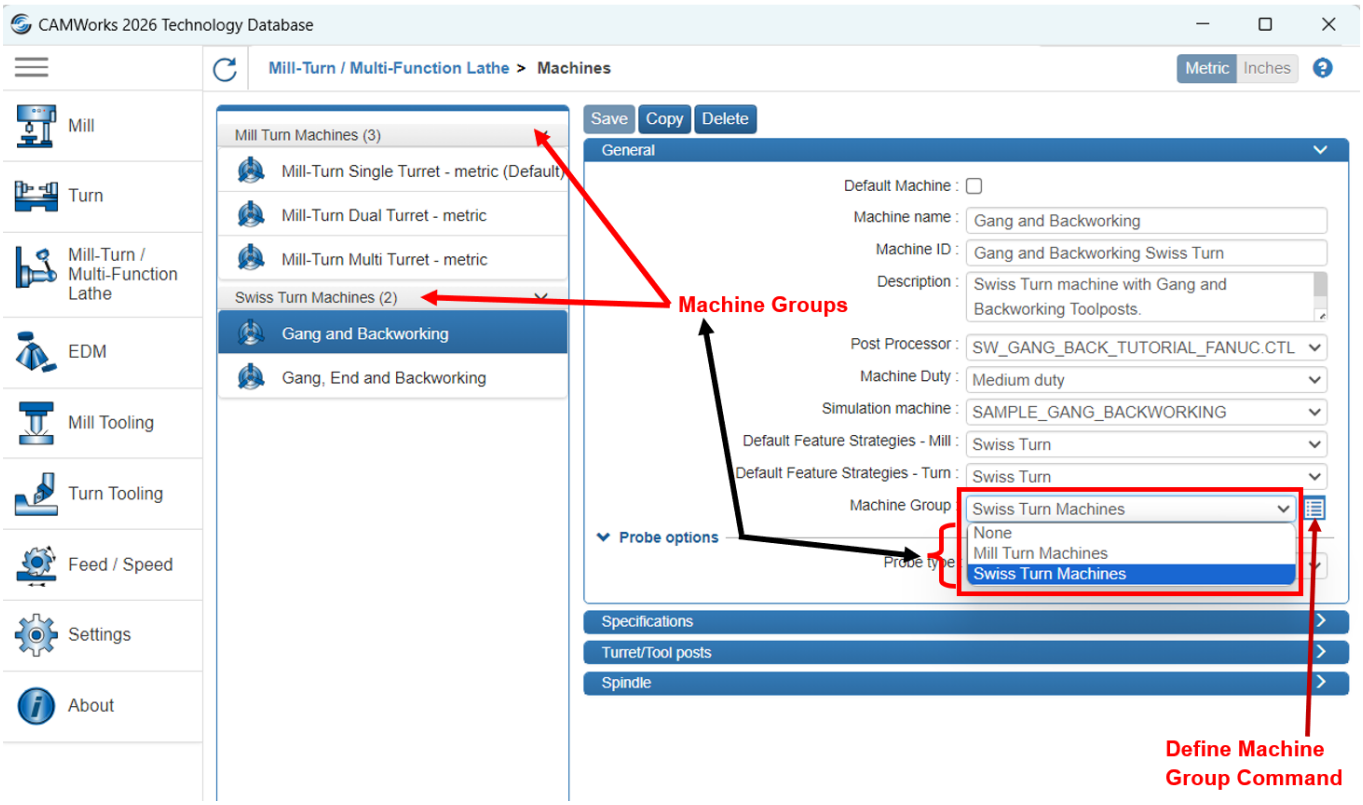
To provide an option to sort the numerous Mill-Turn machines saved within the TechDB into different machine groups for enabling easier identification and classification

Implementation:

The Technology Database allows you to define an unlimited number of Mill-Turn, Swiss Lathes and Multi-Function Lathe machines. To easily identify and classify such machines saved in the ***Mill-Turn/Multi-Function Lathe>>Machines*** form of the TechDB, the option to create Machine groups and assign the saved machines to different Machine Groups has been introduced.

'Machine Group' Dropdown List under General Tab of Machines Form

When a Mill-Turn, Swiss Lathe or Multi-Function Lathe machine available in the *Mill-Turn/ Multi-Function Lathe>Machines* form is selected, its corresponding ***Machine Parameters*** form populated with the *Machine Definition* parameters will be displayed. Under the ***General*** tab of this *Machine Parameters* form, a new dropdown list labeled ***Machine Group*** has been introduced. All the ***Machine Groups*** saved within the TechDB are displayed within this dropdown list. Use this dropdown list to indicate the *Machine Group* that the selected ***Machine*** will be sorted into.



Machine Group Dropdown List under General Tab of Machines Form in Mill-Turn/Multi-Function Lathe> Machines Form

'None' Option in 'Machine Group' Dropdown List under General Tab of Machines Form

Assigning a *Machine Group* to a Mill-Turn machine type is optional. If you do not wish to assign the selected machine to any *Machine Group*, then select the ***None*** option in its *Machine Group* dropdown list.

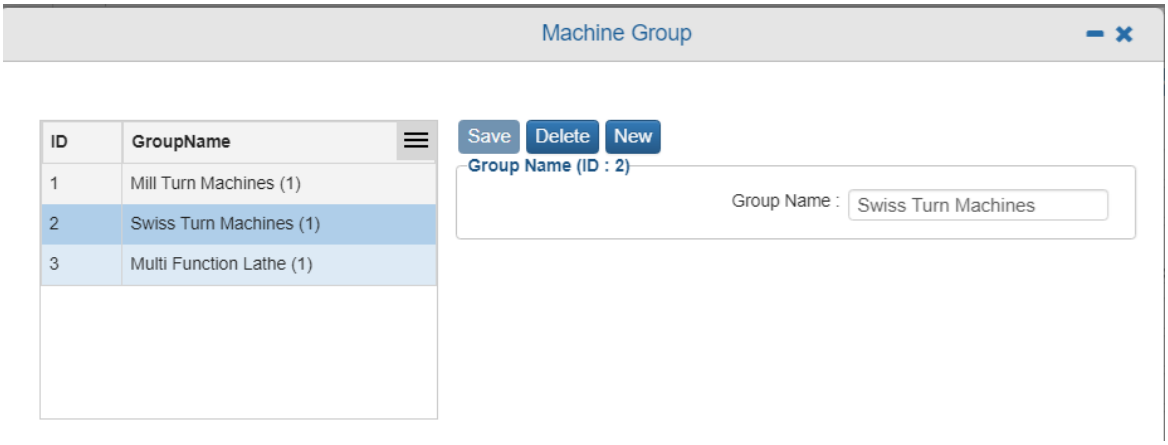
Sorting of Machines based on Machine Groups within the Mill-Turn/Multi-Function Lathe Interface

In both the ***Mill-Turn/Multi-Function Lathe menu*** interface and ***Mill-Turn/Multi-Function Lathe>Machines*** form, all the machines will be sorted and listed based on their respective Machine Groups.
All machines that have *Machine Group* assigned as ***None*** will be listed at the top. All other Machine Groups will be listed after the ***None*** machine group.

The machines under a specific *Machine Group* will be displayed in a collapsible group box. The *Machine Group* will serve as the header of the collapsible group box. A numerical value will be displayed within parentheses adjacent to the header. This number indicates the total number of machines sorted into that specific machine group.

Creating, Editing and Deleting Machine Groups

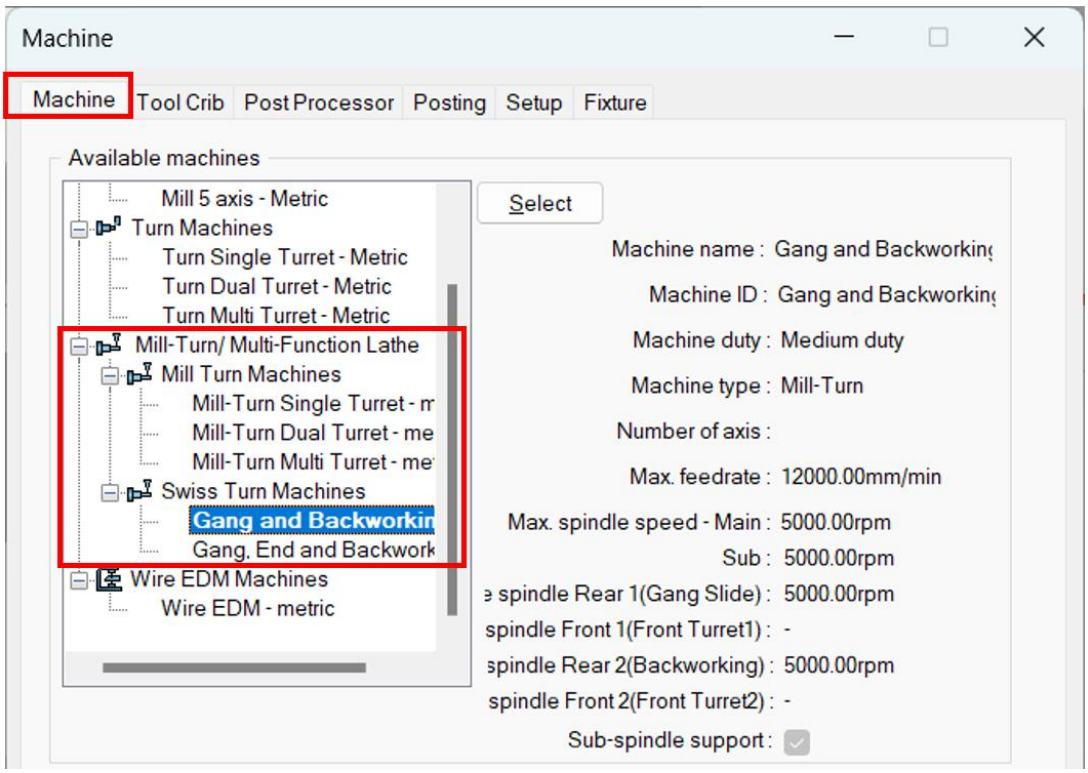
Clicking on the ***Define Machine Group*** button adjacent to the *Machine Group* dropdown list in the ***Machines*** form will display the ***Define Machine Group*** pop-up window. Use this window to view, edit, add, delete, and save machine groups in the TechDB. The machine groups you define will populate the *Machine Group* dropdown list under *General* tab of the *Machines* form within the *Mill-Turn/Multi-Function Lathe>Machines* form.



Machine Group Window to view, create, edit, and/or delete Machine Groups available in TechDB

Machine Groups under Machine Tab of Machine Dialog Box in the CAMWorks Application

In the *CAMWorks* application, all machines of Mill-Turn type will be classified and listed in the ***Available Machines*** list box under the ***Machine*** tab of the ***Machine*** dialog box on the basis of their Machine Group.



Machines of Mill-Turn type classified on the basis of the Machine Group they belong to within Available Machines list box under Machine Tab of Machine Dialog Box

Toolcrib Groups Option to sort and classify Mill-Turn Tool Cribs

Purpose:

To provide an option to sort the numerous Mill-Turn tool cribs saved in the TechDB into different groups so that they can be made available exclusively for selected machines

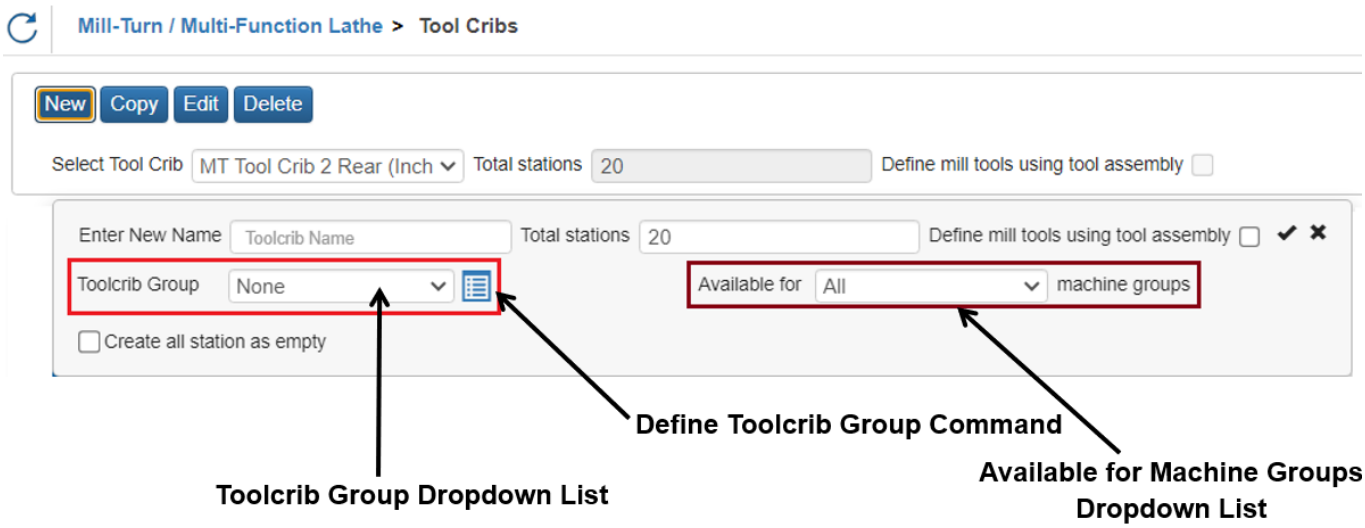
Implementation:

The Technology Database allows you to define an unlimited number of Mill-Turn tool cribs. To easily identify and classify the numerous tool cribs saved in the ***Mill-Turn/Multi-Function Lathe>>Tool Cribs*** form of the TechDB, the following functionalities have been introduced:

- The option to create *Tool Crib groups*
- The option to assign a selected Mill-Turn tool crib saved in the TechDB to a Tool Crib group
- For a selected Mill-Turn tool crib, the option to indicate the specific Machine Group for which it will be available (This will make the tool crib available for assignment to turrets/tool posts of any *Mill-Turn/Multi-Function Lathe* machine belonging to that ***Machine Group***.)

'Toolcrib Group' Dropdown List in Pop-Up Window to Create/Edit Mill-Turn Tool Cribs

In the ***Mill-Turn/Multi-Function Lathe>Tool Cribs*** form within the TechDB, clicking on the ***New*** or ***Edit*** button will display a pop-up window containing parameters associated with the specific tool crib's definition. These parameters are used to create or edit the selected tool crib.



Pop-Up Window to Create/Edit Mill-Turn Tool Cribs

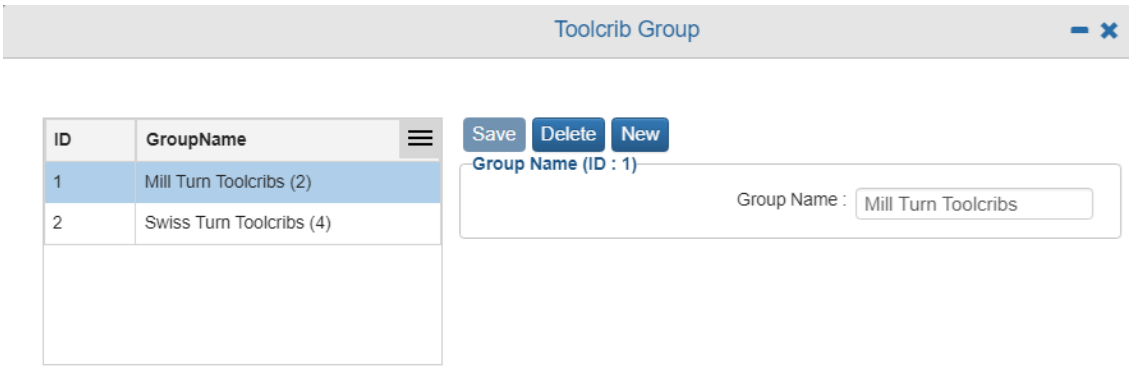
The following new parameters have been introduced in this pop-up window:

- **Tool Crib Group Dropdown List:**

  Use the ***Tool Crib Group*** dropdown list to assign the active tool crib to a specific *Tool Crib Group* available in this dropdown list. Assigning a ***Tool Crib Group*** to any Mill-Turn tool crib saved in the TechDB is optional. Hence, a Tool Crib Group named ***None*** is available in the *Tool Crib Group* dropdown list. If you do not wish to assign the active Tool crib to any ***Tool Crib group***, then select the ***None*** option in the dropdown list. For all Mill-Turn tool cribs that are not sorted into any specific Tool Crib group, the default Tool Crib Group will be *None*.

- **Define Tool Crib Group Command** 

  Clicking on this command button will display the ***Define Tool Crib Group*** pop-up window. Use this pop-up window to create, edit, or delete ***Tool Crib Groups*** available within the TechDB.
  The ***Group Name*** column in this window indicates the names of all the Tool Crib Group as saved within the TechDB. A numerical value will be displayed within parentheses adjacent each tool crib group's name in this column. This value indicates the number of Mill-Turn tool cribs saved in the TechDB that have been assigned to that specific Tool Crib group.
  The *Tool Crib Groups* defined and saved within this window will populate the ***ToolCrib Group*** dropdown list.



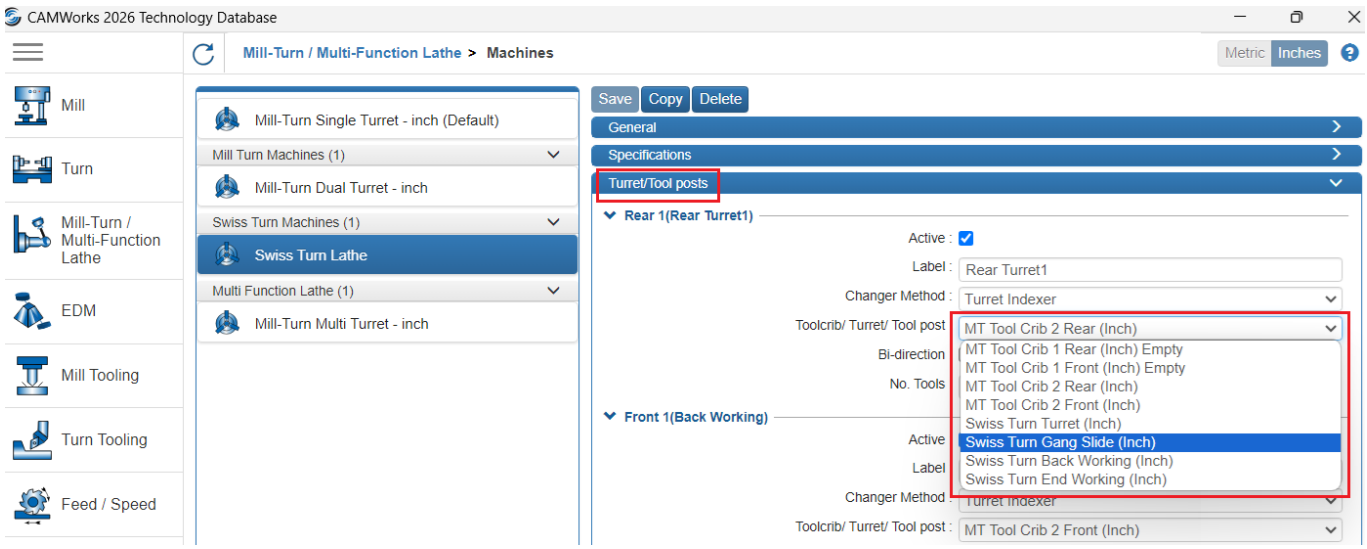
Tool Crib Group Pop-Up Window

- **Available for _____ machine groups (Dropdown List)**

  Use this dropdown list to indicate the Machine group for which the active Mill-turn tool crib will be available for selection.
  If you wish to make the active tool crib available for all machines/machine groups, then select the option ***All*** within this dropdown list.

Turret/Tool Posts Tab for Mill-Turn Machines

For *Mill-Turn/Multi-Function Lathe* machines, information associated with its tool cribs, turrets and/or Tool Posts can be viewed, defined, and edited under the ***Turret/Tool Posts*** tab of the *Mill-Turn/Multi-Function Lathe>Machines* form of the Technology Database. Use the ***Turret/ Tool Post*** dropdown list present under this tab to select the default tool crib to be assigned for the selected turret/Tool post of the machine being edited. This dropdown list will be populated with tool cribs defined in the *Tool Cribs* form of the TechDB. However, only tool cribs that have been assigned to the same Machine Group as that of the machine being edited will be displayed in this dropdown list.

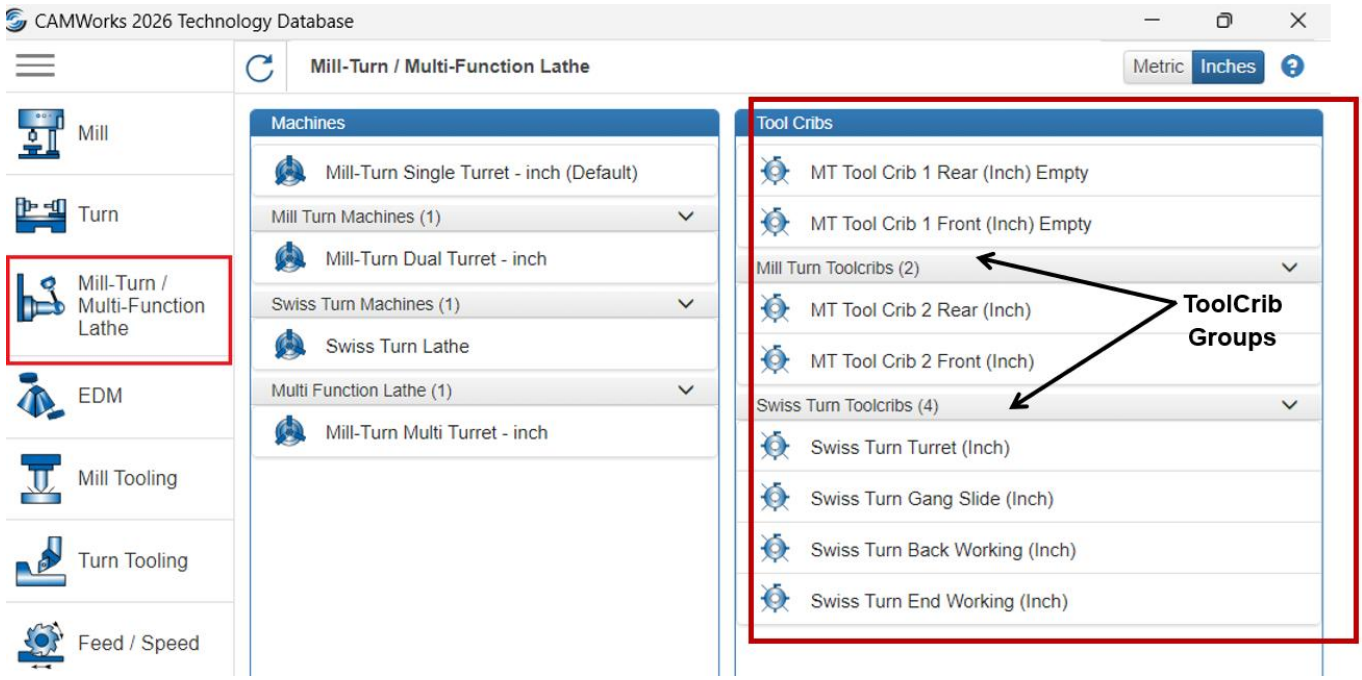


Turret/Tool Post Dropdown List under Turret/Tool Posts Tab of Machines Form in Mill-Turn/Multi-Function Lathe> Machines Form

Viewing Tool Crib Groups on the Mill-Turn/Multi-Function Lathe Form

In the *Mill-Turn/Multi-Function Lathe* menu user interface, all the Mill-Turn tool cribs within the TechDB are listed under the Tool Cribs sub-menu. In this user interface, all the Tool Cribs saved within the TechDB that are not sorted into Tool Crib group will be displayed at the top of the list box. (These are the Tool Cribs that have the Tool Crib Group property assigned as *None*. After these non-sorted tool cribs, the Tool Crib Groups will be listed in alphabetical order as headers of collapsible groups. The tool cribs sorted into a specific Tool Crib group will be listed under that Tool Crib group in the form of a collapsible box.

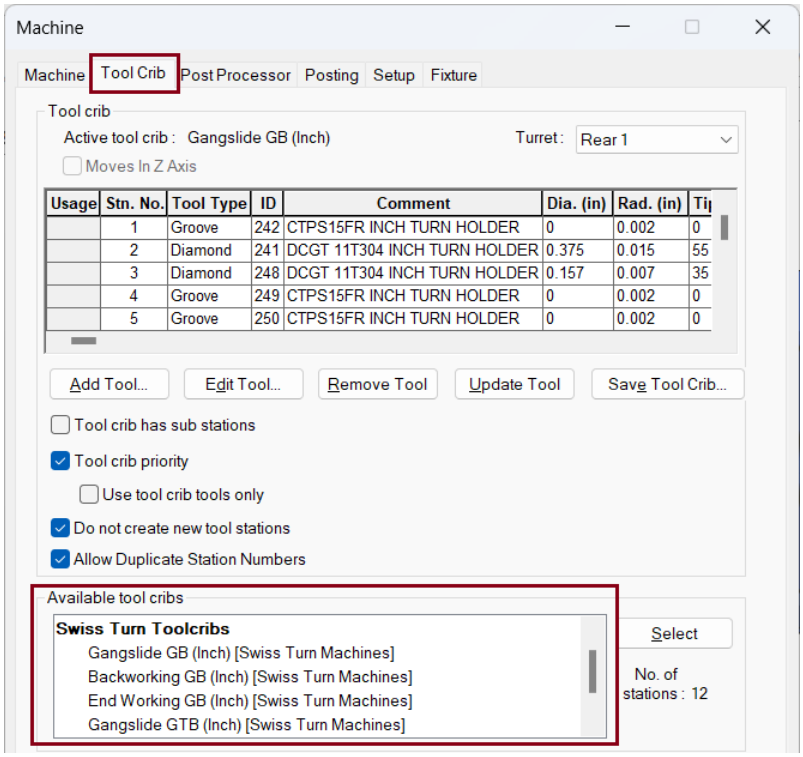
A numerical value will be displayed within parentheses adjacent to each header. This number indicates the total number of tool cribs sorted into that specific Tool Crib group.



Mill-Turn Tool Cribs listed under Tool Cribs Groups they belong to in the Mill-Turn/Multi-Function Lathe Menu User Interface

Available Mill-Turn Tool Cribs listed as per Tool Crib Groups in CAMWorks

In the *CAMWorks* application, when a Mill-Turn/Multi-Function Lathe machine is selected as the active machine, the Available Tool Cribs list box under the ***Tool Crib*** tab of the ***Machine*** dialog box will display and sort all the available Mill-Turn Tool Cribs on the basis of the *Tool Crib Group* assigned to it. All the Mill-Turn Tool Cribs assigned a specific Tool Crib Group will be displayed under a label bearing that *Tool Crib Group's* name. All those Mill-Turn Tool Cribs that have not been assigned any *Tool Crib Group* will be at the top of the ***Available Tool Cribs*** list box.



Available Tool Cribs List Box under Tool Crib Tab of Machine Dialog Box

Collets as Work Holding Device for Turn and Mill-Turn Machines

Purpose:

To provide an option to define and assign a Collet as work holding device for the Main and/or Sub Spindles of Turn and Mill-Turn machines

Implementation:

As a clamping device, collets are capable of producing a high clamping force and accurate alignment.

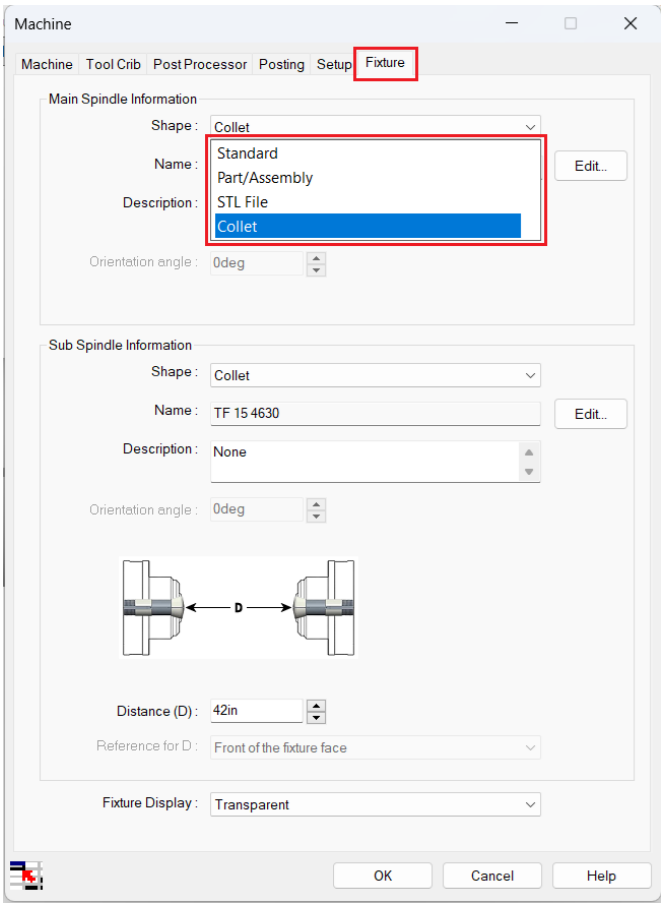
In previous versions of *CAMWorks*, in Turn and Mill-Turn mode, only the *Standard* and user-defined chucks/fixtures were supported as work holding devices for Turn and Mill-Turn machines.

From *CAMWorks 2025* version onwards, an option to define a Collet as the work holding device has been provided. *CAMWorks* will allow users to define and maintain a centralized library of collets in Technology Database (TechDB). The user will be able to assign a desired collet saved in the TechDB to the Turn/Mill-Turn machine selected for machining a part model/ assembly.

Sample Collet Image

Machine Dialog Box

In the Machine dialog box, the *Chuck/Fixture* tab has been renamed to *Fixture* tab. The *Shape* dropdown list within the *Main Spindle Information* and *Sub Spindle Information* under this tab is used to determine the type of work holding device. A new option *Collet* has been added to this dropdown list. Once this option is selected, the name of the collet assigned to the machine will be displayed in the *Name* field below the *Shape* dropdown list. (This default selection is based on the default collet assigned to the Machine in the TechDB.)

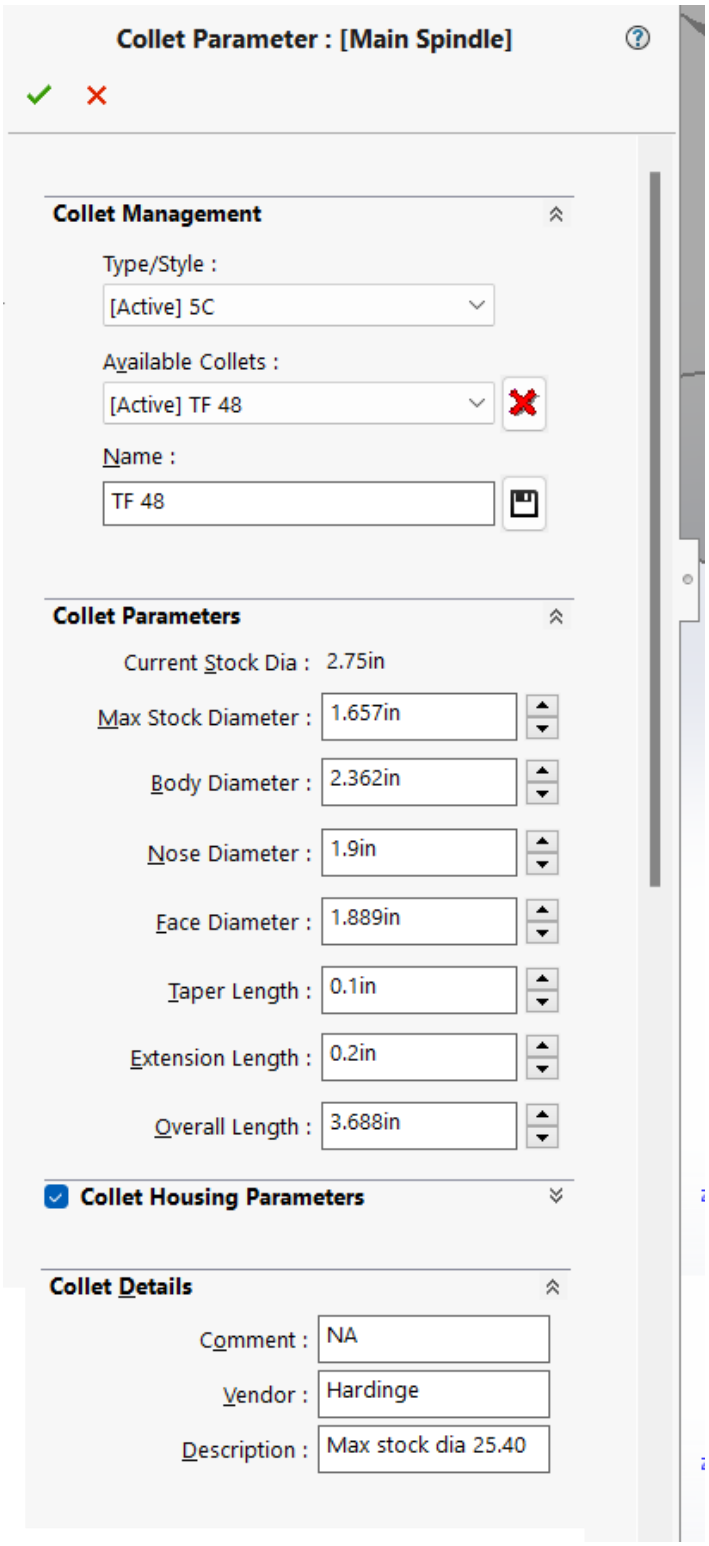


Collet option within the Shape dropdown list under Fixture tab of Machine dialog box

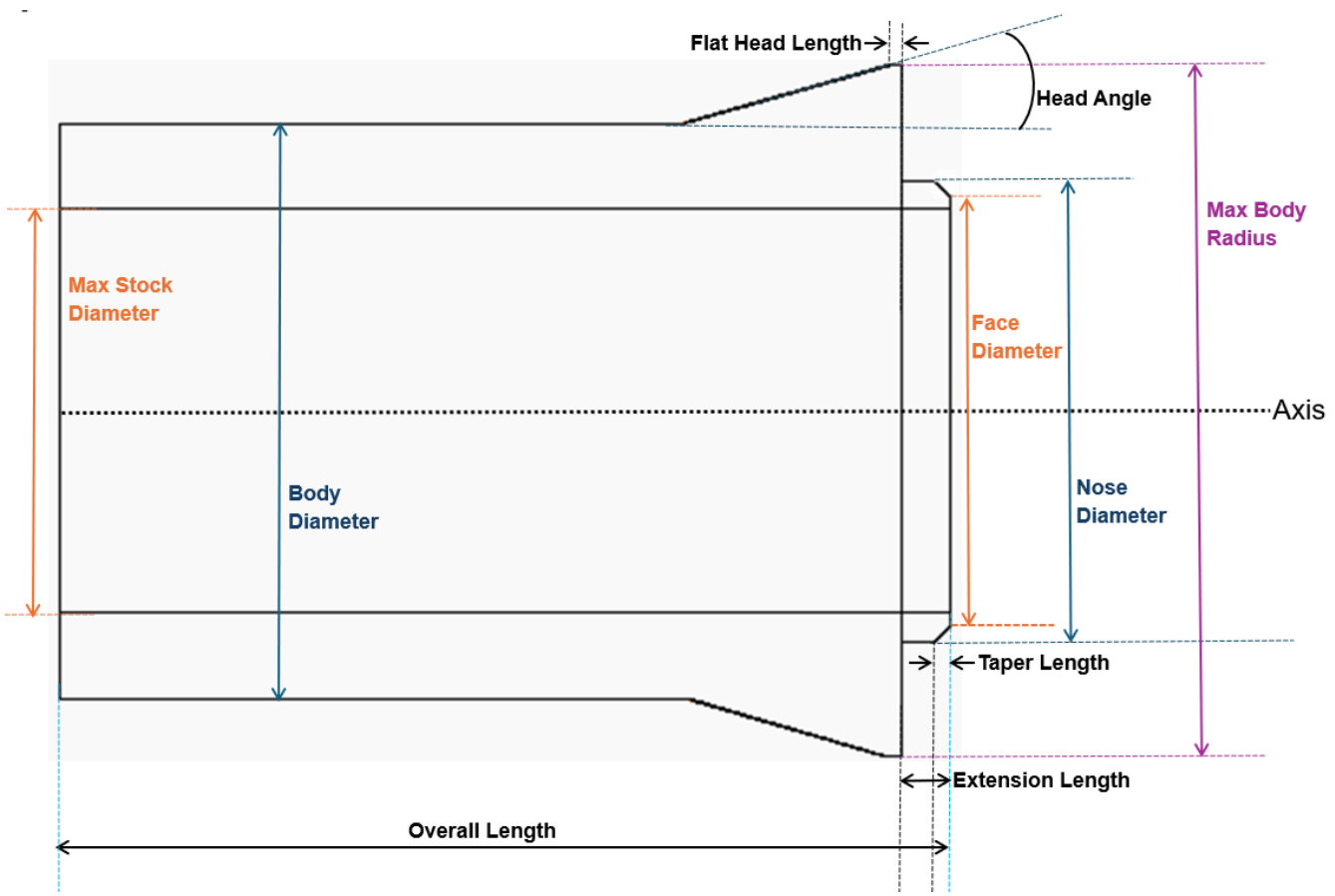
Collet Parameter Dialog Box

If you wish to assign another collet as the work holding device, then click on the ***Edit*** button adjacent to the *Name* dropdown list under the ***Fixture*** tab. Clicking on this command button will display the ***Collet Parameter: Main/Sub Spindle Dialog Box***. Use the parameters and options available within this dialog box to:

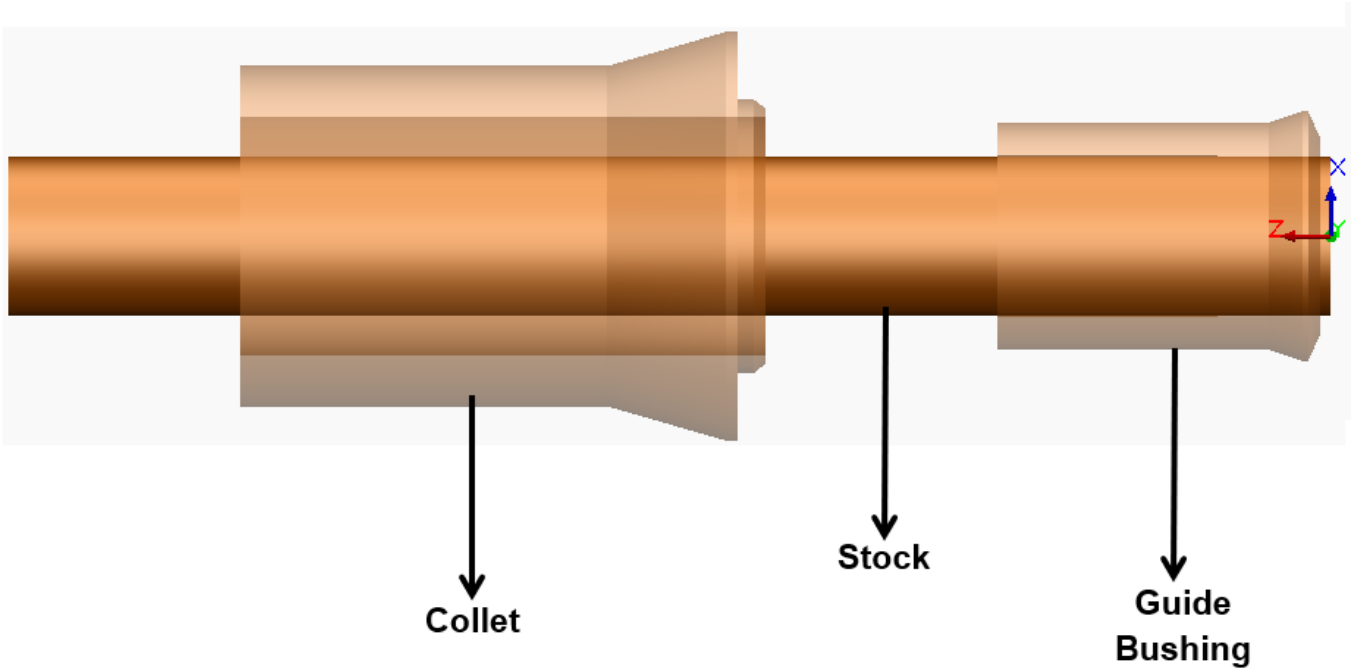
- Change the collet type/style and corresponding collet selection
- Create new collets and optionally save them to the TechDB
- Edit the parameters of the selected collet



Collet Parameter: Main/Sub Spindle dialog box displayed when Edit button is clicked



Geometry of the Collet Profile

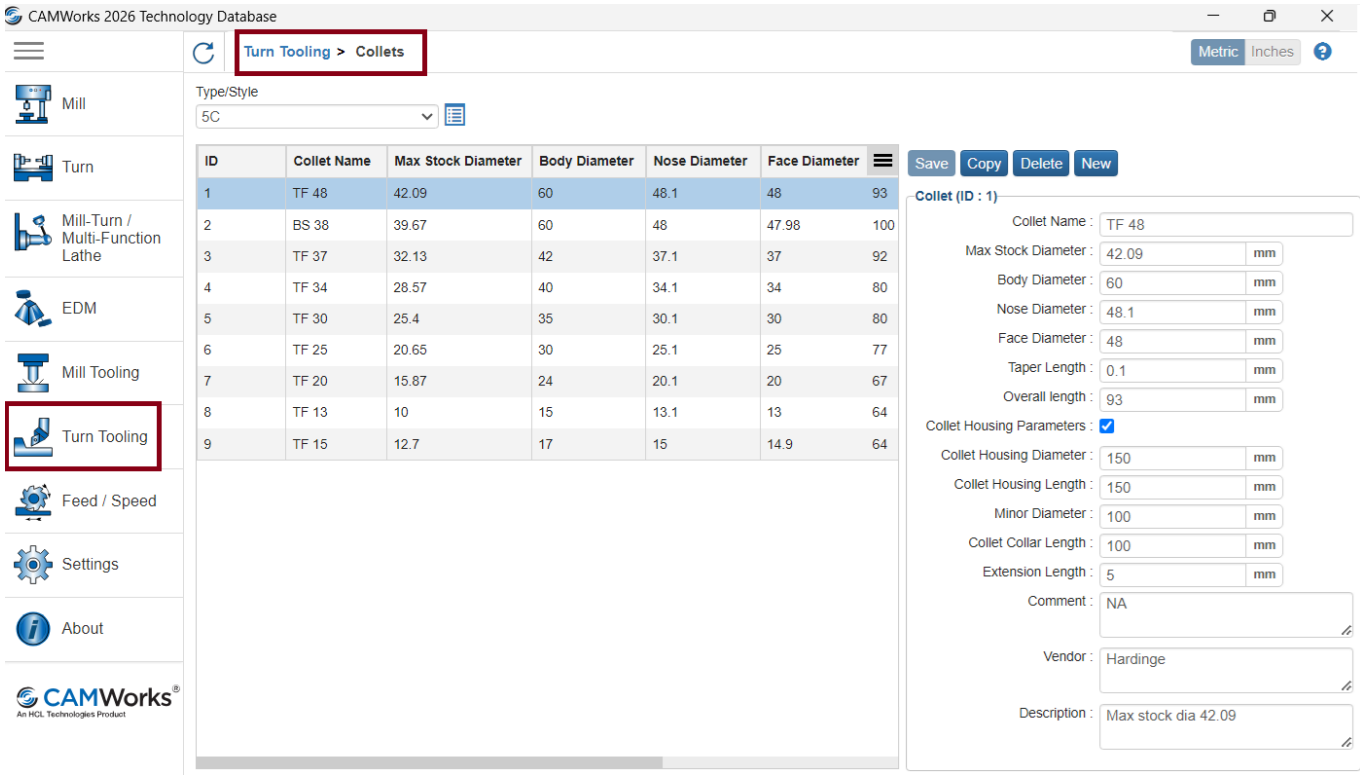


Sample Image of a Collet along with the Stock and Guide Bushing

Collets Form in TechDB

Within the TechDB, to invoke the ***Collets*** form, click on the *Collets* menu item under the ***Holders & Assemblies*** sub-menu of the Turn tooling menu.

The ***Collets*** form allows you to create a centralized library of collets. All collets saved within the TechDB are displayed in a tabular format within the *Collets* form. Saving collets in TechDB using this form eliminates the need to define collets for individual Turn and Mill-Turn/Multi-Function Lathe machines. It thereby allows you to automate the selection of collets as a work holding device for Turn and Mill-Turn/Multi-Function Lathe machines in CAMWorks.



Collets Form in Technology Database

Option to Define Collet Housing Parameters

Purpose:

To provide the option to define basic parameters of a Collet Housing so that it can be visualized while programming a part/assembly

Implementation:

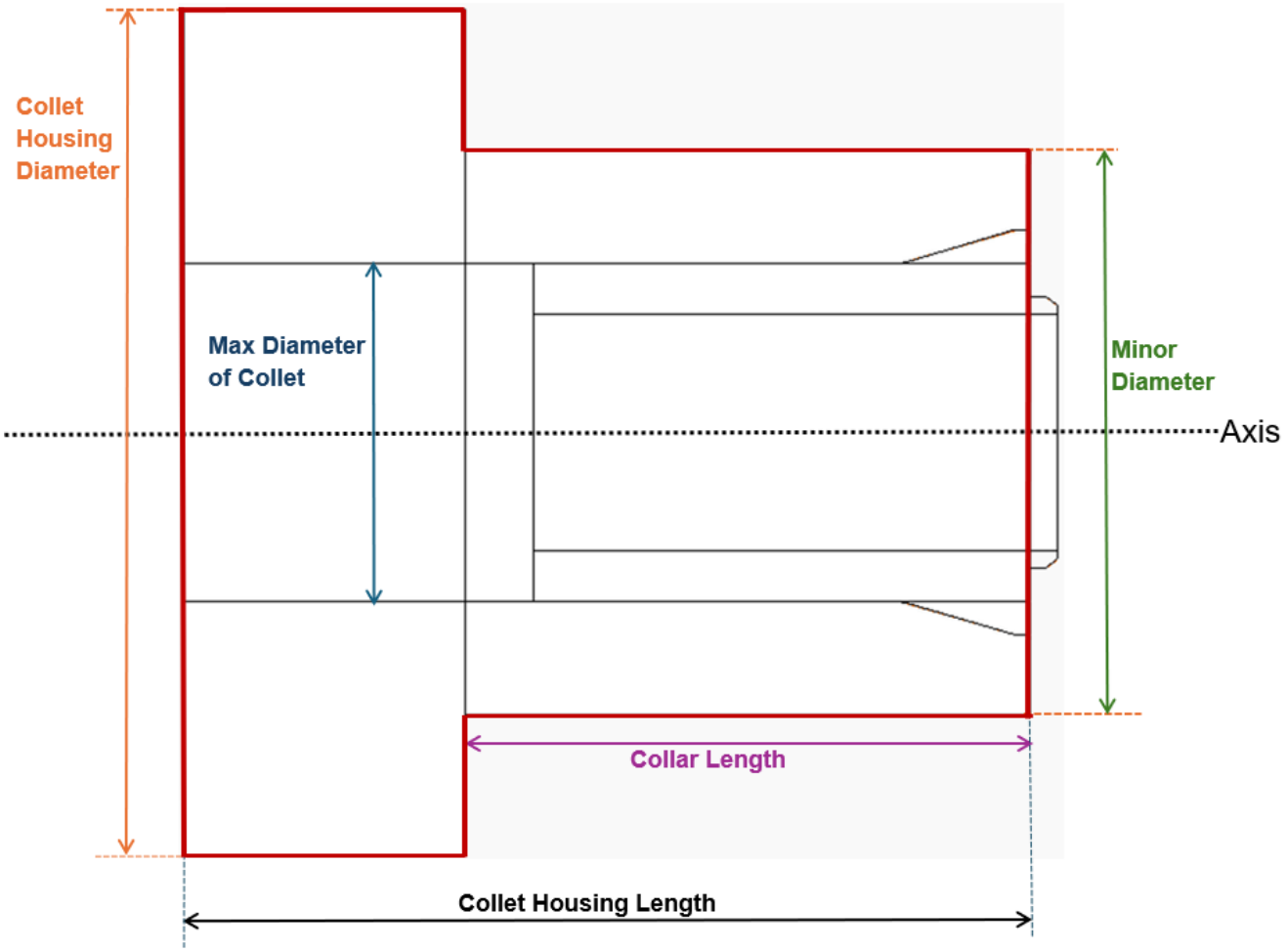
'Collet Housing' refers to the clamping fixture used to hold the collets. In the previous version of CAMWorks, no options were available to define a Collet Housing.

From *CAMWorks 2026* version onwards, a new group box labelled *Collet Housing Parameters* has been introduced within the *Collet Parameters* dialog box. Use the parameters available within this group box to define the collet housing parameters. Providing the definition for collet housing allows you to define and visualize the geometry of the collet housing directly within the graphics area.

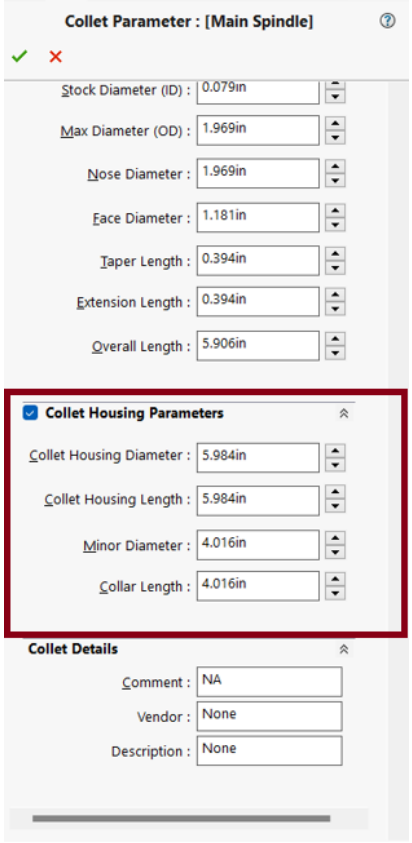
Collet Housing Group Box within Collet Parameters Dialog Box

The following parameters are made available in *CAMWorks 2026* to define collet housing:

- **Collet Housing Diameter:** Allows users to view, assign, or edit the largest diameter of the collet housing.
- **Collet Housing Length:** Enables users to view, assign, or edit the overall length of the collet housing.
- **Minor Diameter:** Users can view, assign, or edit the diameter of the front diameter section of the collet housing.
- **Collar Length:** Allows users to view, assign, or edit the length of the front diameter section of the collet housing.



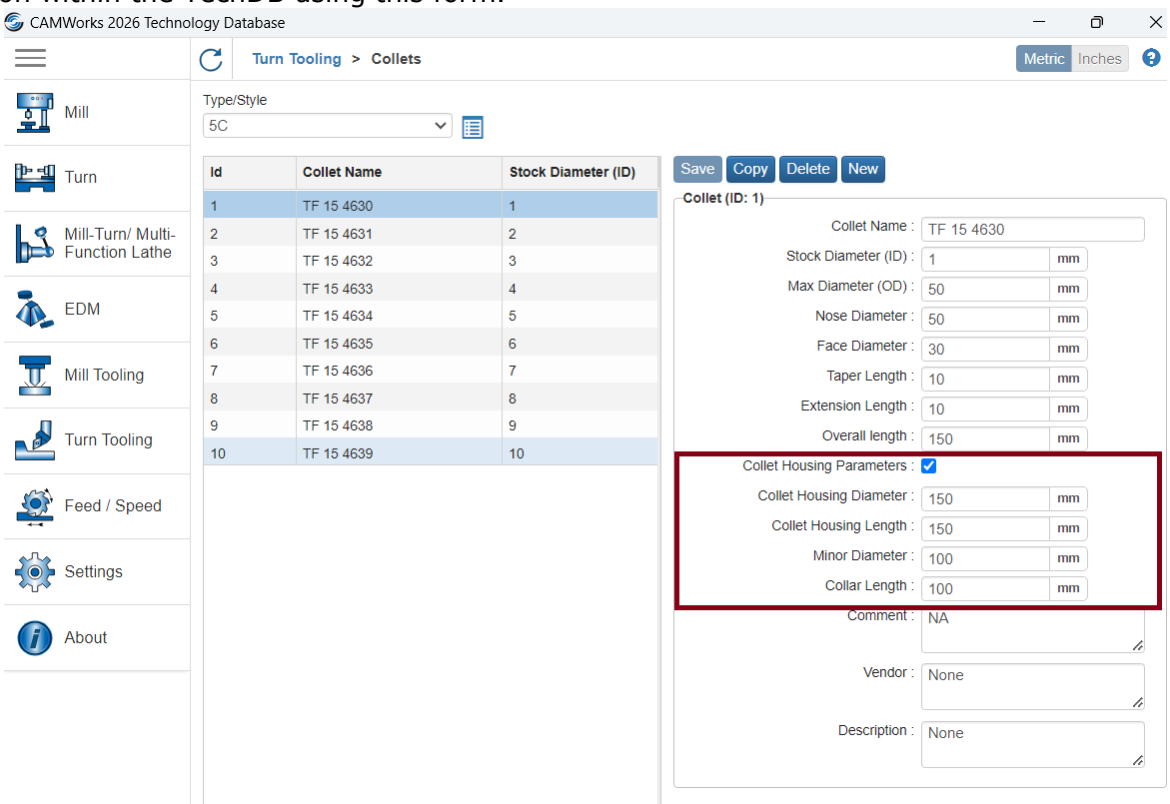
Geometry of the Collet Housing Profile



Collet Housing Parameters Group Box in Collet Parameters Dialog Box

Saving Default Collet Housing Parameters in TechDB

The parameters associated with defining *Collet Housing* are available in the ***Collets*** form provided under Turn Tooling menu of the TechDB. You can assign the default values for *Collet Housing* definition within the TechDB using this form.



Collet Housing Parameters in Collets Form available under Turn Tooling Menu of TechDB

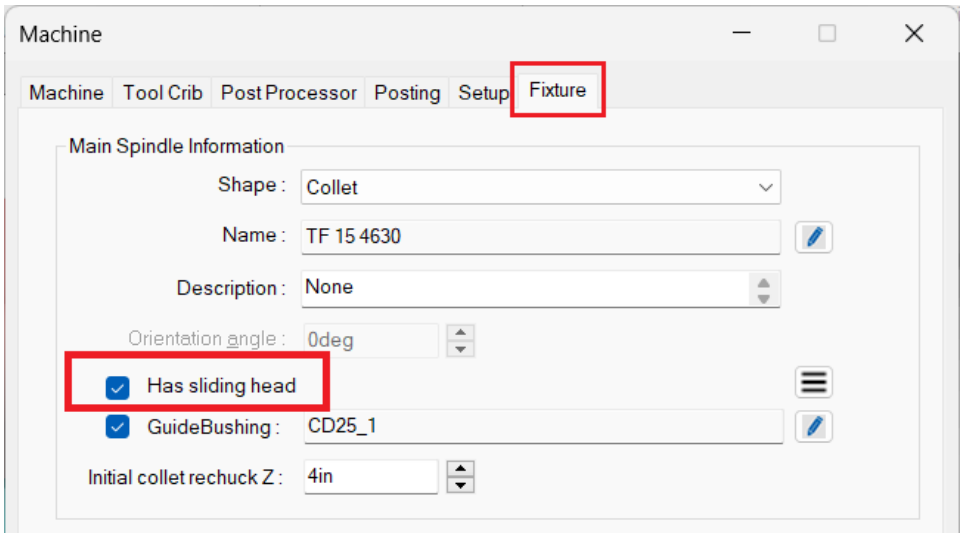
Activating Sliding Head Option for a Selected Machine

Purpose:

To indicate that the active Mill-Turn/Multi-Function Lathe machine selected has a sliding head so that tools are assigned accordingly to all the operations ensuring that the generated toolpaths suit the sliding head machines and post processed accordingly

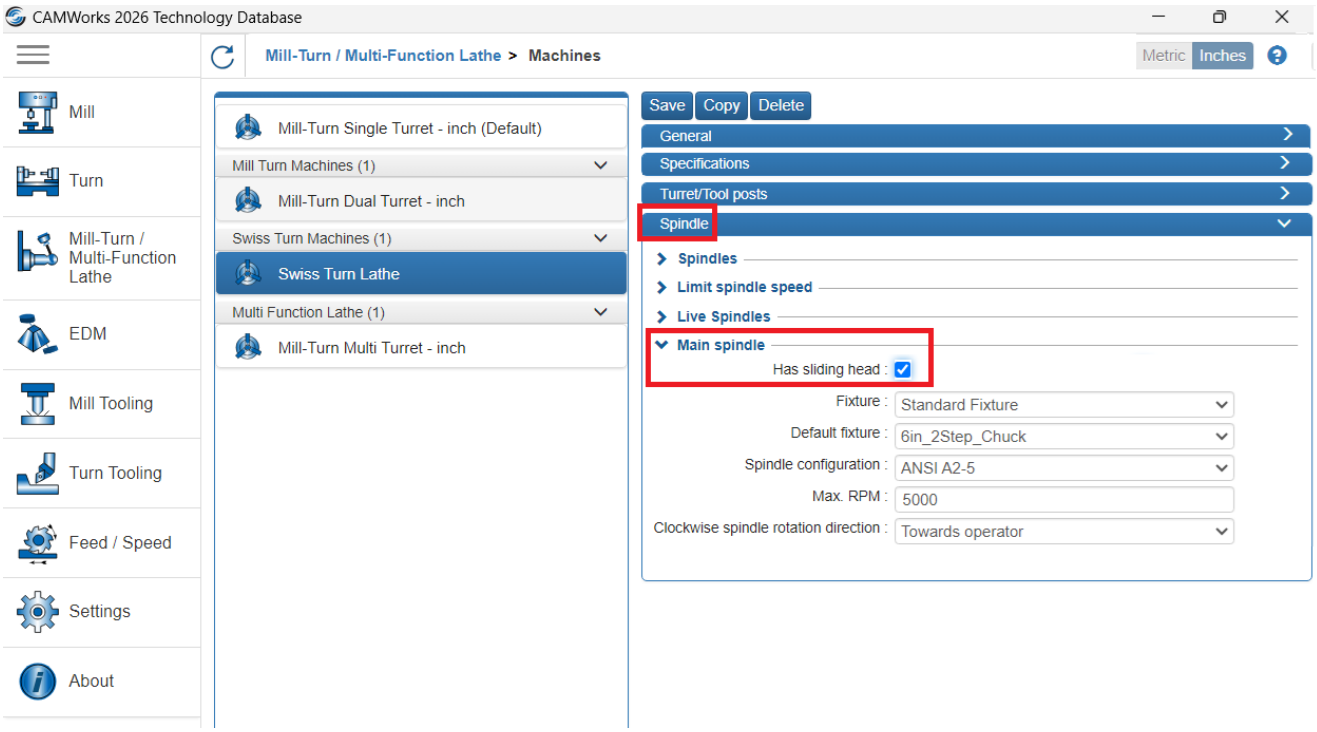
Implementation:

The ***Has sliding head*** checkbox option is present under the ***Fixture*** tab of the Machine dialog box when the active machine selected is a ***Mill-Turn/Multi-Function Lathe*** machines. Use this checkbox to indicate whether the machine is a traditional fixed head Mill-Turn machine or a sliding head lathe. When checked, it indicates that the active machine has a sliding head.



'Has sliding head' Checkbox Option under Fixture Tab of the Machine Dialog Box

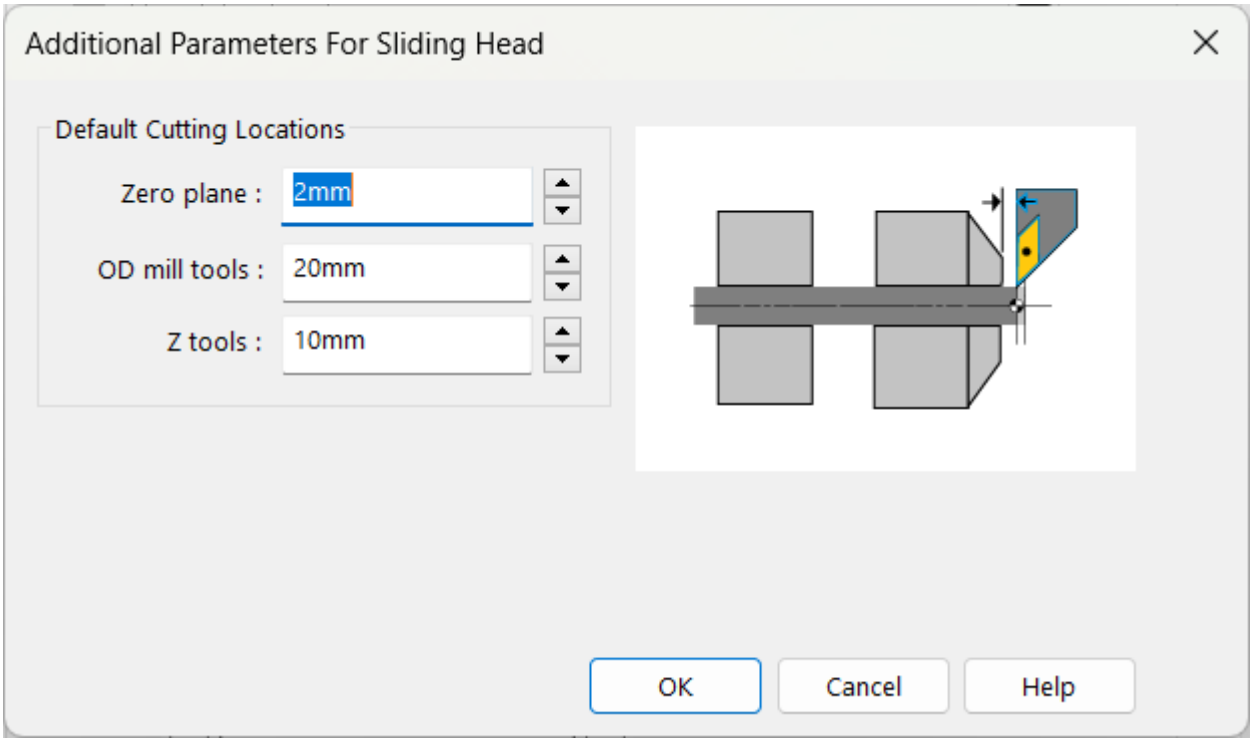
The default setting for this checkbox option can be assigned to the *Has sliding head* checkbox parameter present in the ***Spindle*** Tab under ***Mill-Turn/Multi-Function Lathe>Machines*** form in the Technology Database.



Option to Assign Default Setting for 'Has Sliding Head' under Spindle Tab for a Mill-Turn/Multi-Function Lathe Machine Defined and Saved within TechDB

Visualizing Tools in Graphics Area at Designated Distances For Machines with Sliding Heads

While simulating the toolpaths on a Mill-Turn/Multi-Function Lathe Machine with a sliding head, the tools used for machining will be visualized at a designated location. Clicking on the ***Additional Parameters For Sliding Head*** command button adjacent to the ***Has sliding head*** checkbox will display the ***Additional Parameters For Sliding Head*** dialog box. Use this dialog box to assign/edit these default cutting locations (distances) as per the specifications required for the active machine.



'Additional Parameters For Sliding Head' Dialog Box

- **Zero Plane**: Use this parameter to indicate the distance from the machine origin of the part to the face of the Guide Bushing.
- **OD Mill Tools**: Use this parameter to indicate the distance from the machine origin to the centerline of Mill tools that will be used to machine on the OD of the part.
- **Z Tools**: Use this parameter to indicate the distance from the machine origin to the tip of mill tools that will be used to machine the part from the Z direction.

Guide Bushings when Machining with Swiss Turn/Multi-Function Lathe Machines

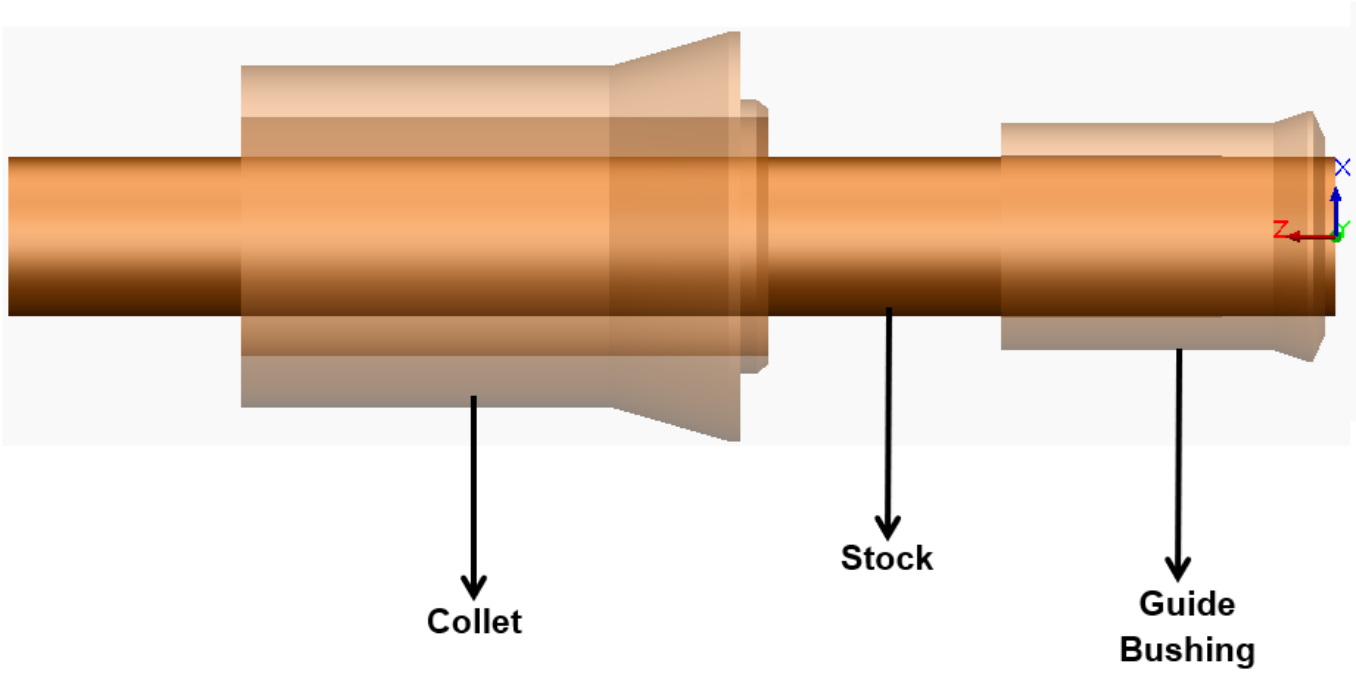
Purpose:

Support for Guide Bushings to enable machining with Swiss Turn/Multi-Function Lathe Machines

Implementation:

What are Guide Bushings?

Guide Bushings are a critical component of Swiss Machines and serve as a major factor differentiating them from conventional CNC Mill-Turn machines. They function as supporting devices for bar stock to be machined on Swiss Turn machines. A clamped bar stock can move axially through the guide bushings. Such a move ensures that the stock is rigidly supported at all times and the turning tool remains closer to the guide bushing. This enables the machining of parts with high quality finishes and extreme accuracies.



Sample Image Displaying Guide Bushing

Guide Bushing Dialog Box in CAMWorks 2026

As *Guide Bushings* are integral to machining with Swiss Turn Lathes/Multi-Function Lathes with sliding heads, user interfaces for viewing, editing, adding, and assigning Guide Bushings have been introduced in *CAMWorks*.

Enabling Guide Bushing within CAMWorks for a Selected Machine

Under the Fixture tab of the Machine dialog box, a checkbox labeled Guide Bushing will be displayed when the following conditions are fulfilled:

- The active machine is a Mill-Turn/Swiss Turn/Multi-Function Lathe machine
- The ***Collet*** option is selected in the ***Shape*** dropdown list under the ***Fixture*** tab of the ***Machine*** dialog box
- The ***Has sliding head*** checkbox option under the ***Fixture*** tab of the ***Machine*** dialog box is checked

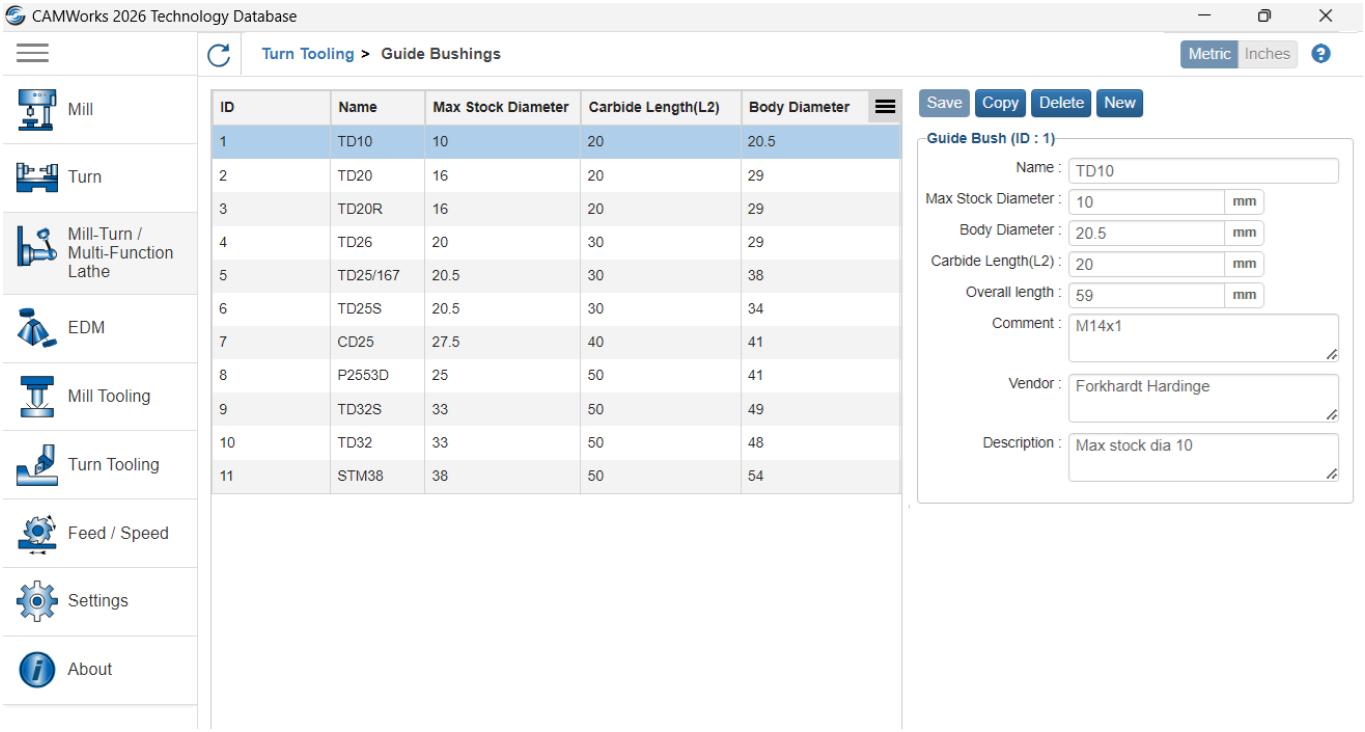
Place a check in the ***Guide Bushing*** checkbox if you wish to enable guide bushing for the selected machine.

Name of Selected Guide Bushing

The ***Name*** field adjacent to the ***Guide Bushing*** checkbox displays the name of the Guide Bushing currently used for the active machine. This is a read-only field. When the Guide Bushing checkbox is checked, the name of the first Guide Bushing saved in the library of Guide bushings within the TechDB will be displayed in this field.

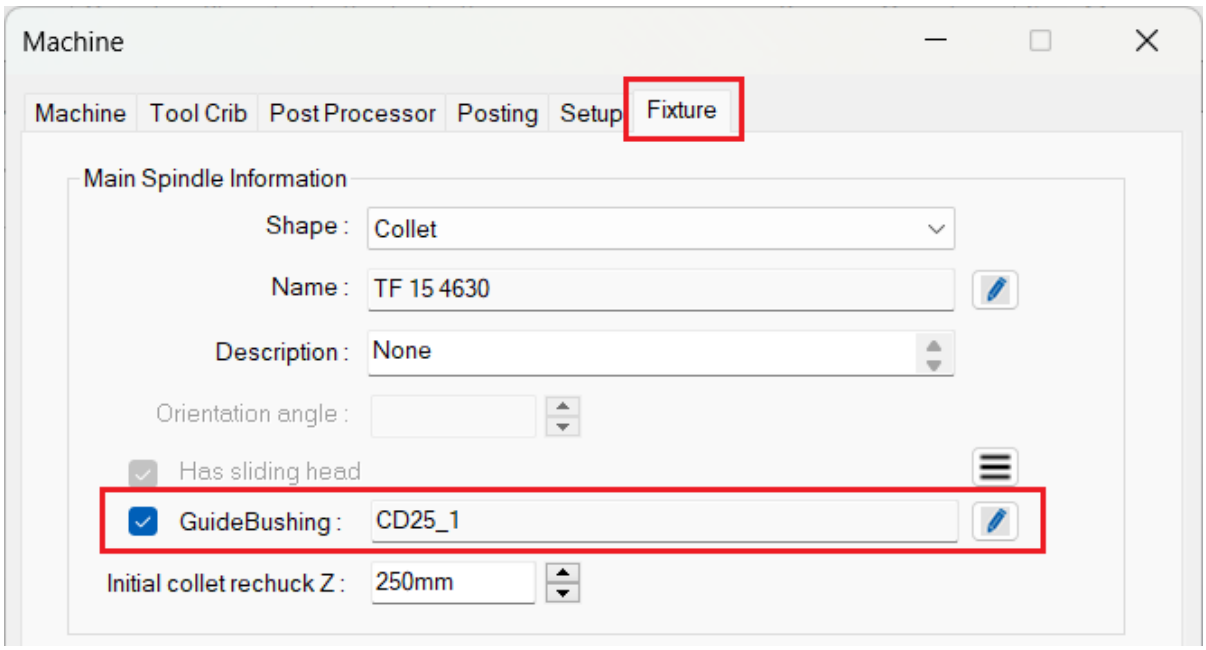
Library of Guide Bushing Saved in Technology Database

Within the Technology Database, browse to the Turn Tooling menu. Clicking on the Guide Bushings menu item under the Holders, Assemblies and Collets sub-menu will display the ***Guide Bushings*** form. Use the parameters available within this form to add, edit, copy, and/or delete Guide Bushings to be used with your Swiss Machines/Multi-Function Lathe machines.



Guide Bushings User Inerface available under 'Turn Tooling>Holder, Assemlies and Collets' Menu of Technology Database (TechDB)

Fixture Tab under Machine Dialog Box



Settings for Guide Bushing under Fixture Tab of the Machine Dialog Box

Invoking the 'Guide Bushing' Dialog Box using Edit Button

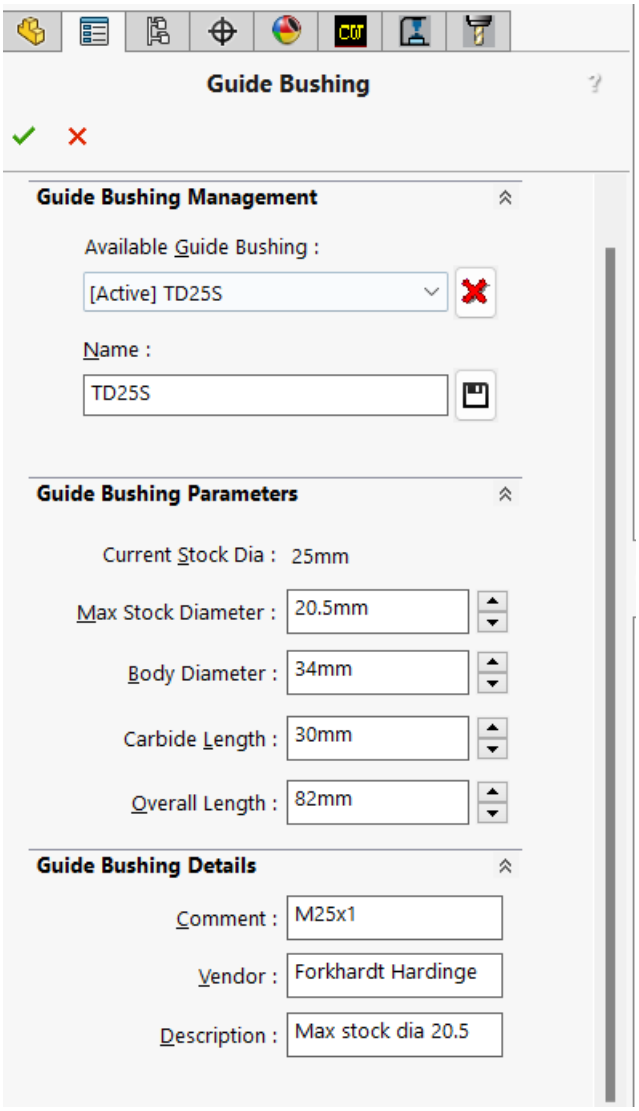
If you wish to change this default Guide Bushing that has been assigned, then click on the ***Edit...***

Button adjacent to this field.

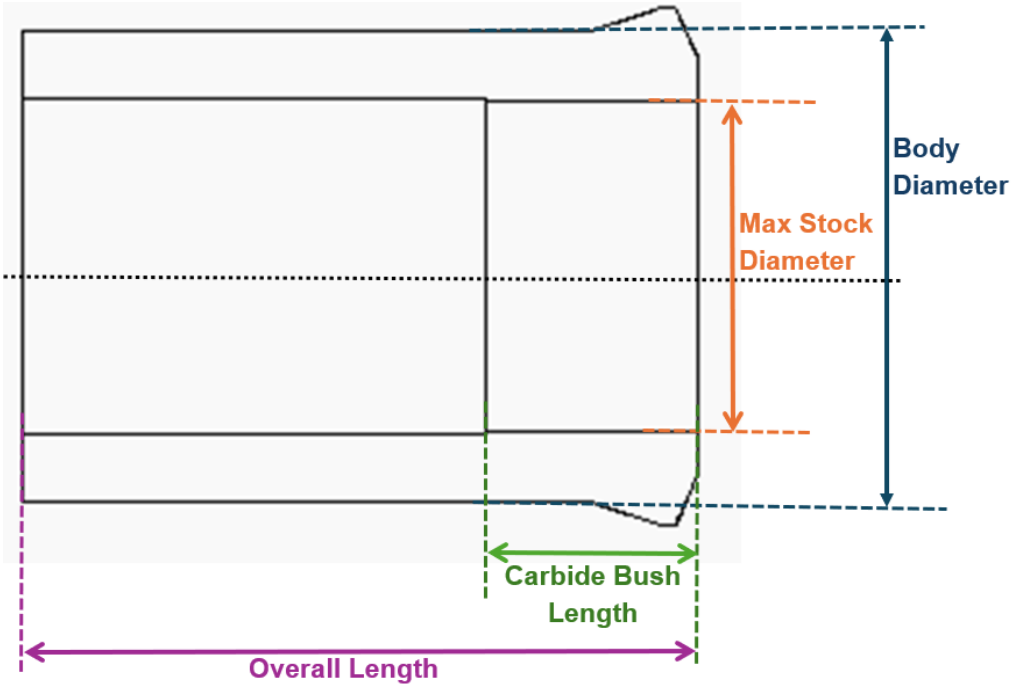
Use the displayed ***Guide Bushing*** dialog box to select another Guide Bushing or edit parameters of a selected Guide Bushing or to add and assign another Guide Bushing.

The ***Available Guide Bushing*** dropdown list in this dialog box will be populated with Guide Bushings saved in the Technology Database.

Once the ***Guide Bushing*** dialog box is closed and the interface reverts to the ***Fixture*** tab of ***Machine*** dialog box, the ***Name*** field will display the name of the newly selected Guide Bushing.



Guide Bushing Dialog Box



Geometry of the Guide Bushing Profile

'Initial collet rechuck Z' to indicate distance between Collet and Part Origin

Purpose:

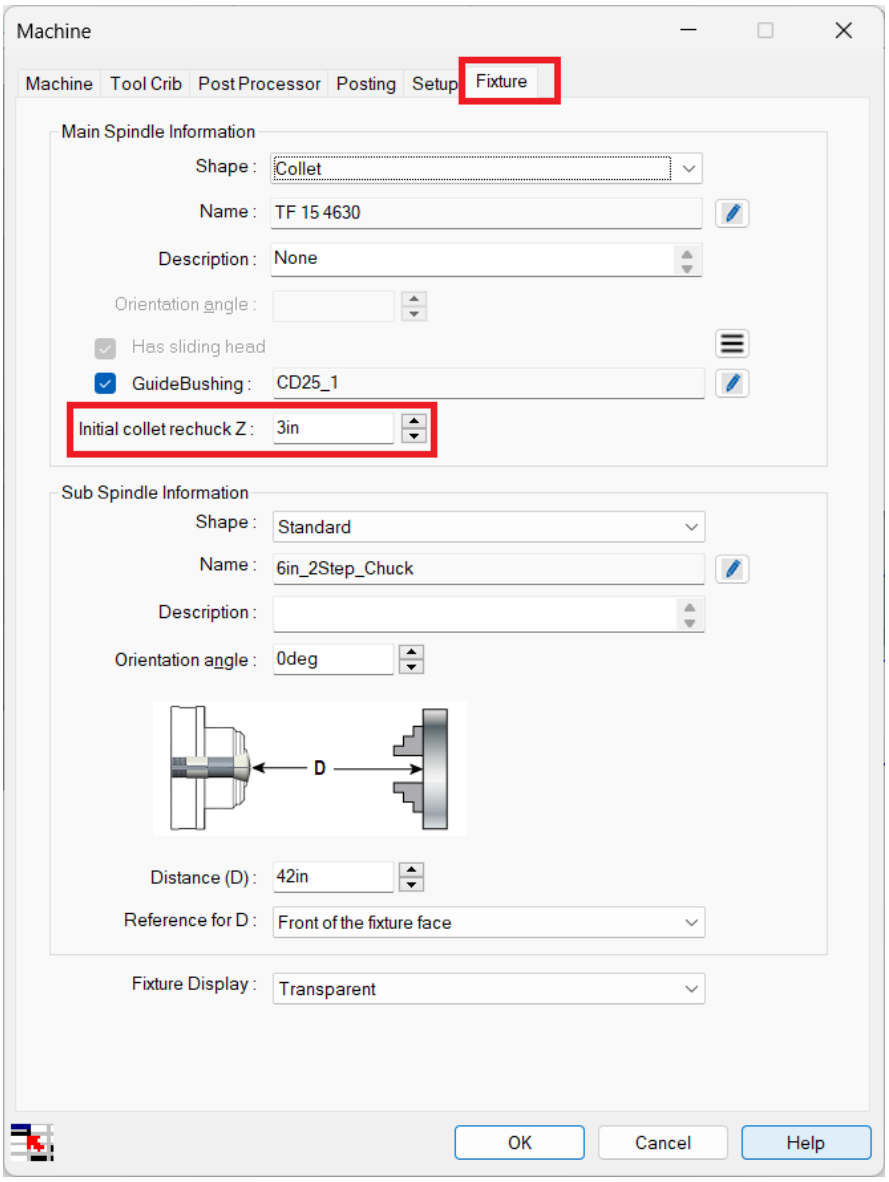
Parameter to indicate the distance from the face of the collet to the origin of the part to be machined when a collet is used to clamp the stock being machined on a Mill-Turn/Multi-Function Lathe machine with sliding head

Implementation:

From CAMWorks 2026 version onwards, when a Mill-Turn/Multi-Function Lathe machine with a sliding head is selected in the *Machine* dialog box, a new parameter ***Initial collet rechuck Z*** will be displayed under the ***Fixture*** tab of the *Machine* dialog box.

The ***Initial collet rechuck Z*** parametric field will be enabled when the following conditions are fulfilled:

- The active machine is a Mill-Turn/Multi-Function Lathe machine
- The ***Collet*** option is selected in ***Shape*** dropdown list under *Fixture* tab of *Machine* dialog box.
- The ***Has sliding head*** checkbox option under the *Fixture* tab of *Machine* dialog box is checked.



The 'Initial collet rechuck Z' Parametric Field under Fixture Tab of Machine Dialog Box

Use this parametric field to indicate the distance from the face of the collet to the origin of the part (Main Spindle Coordinate System). The value you assign to this parameter must be either zero or a value greater than zero. The distance value must be set such that the collet doesn't collide with the *Guide Bushing* during the machining of the part.

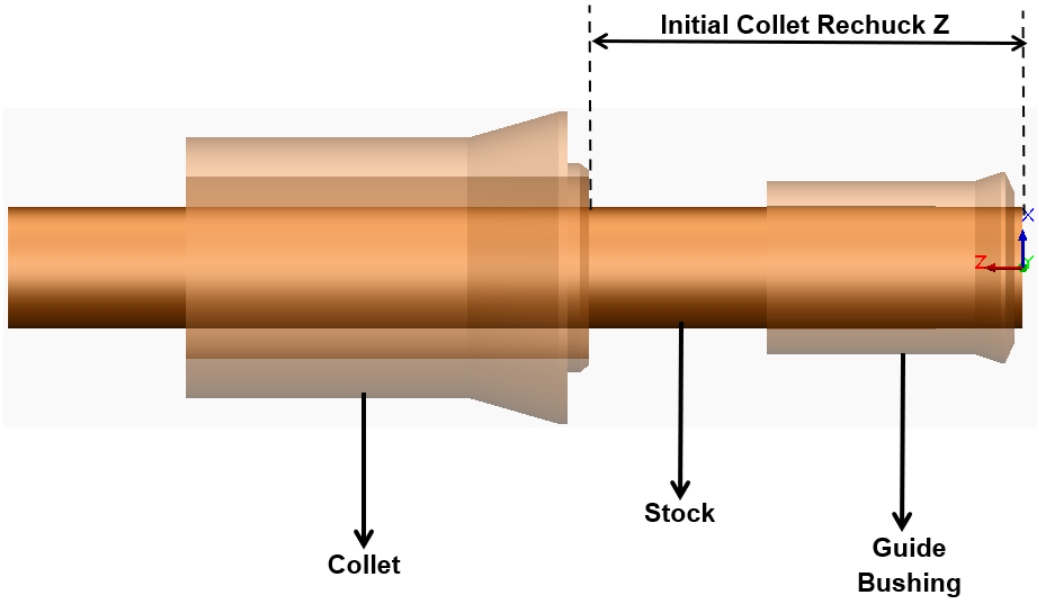
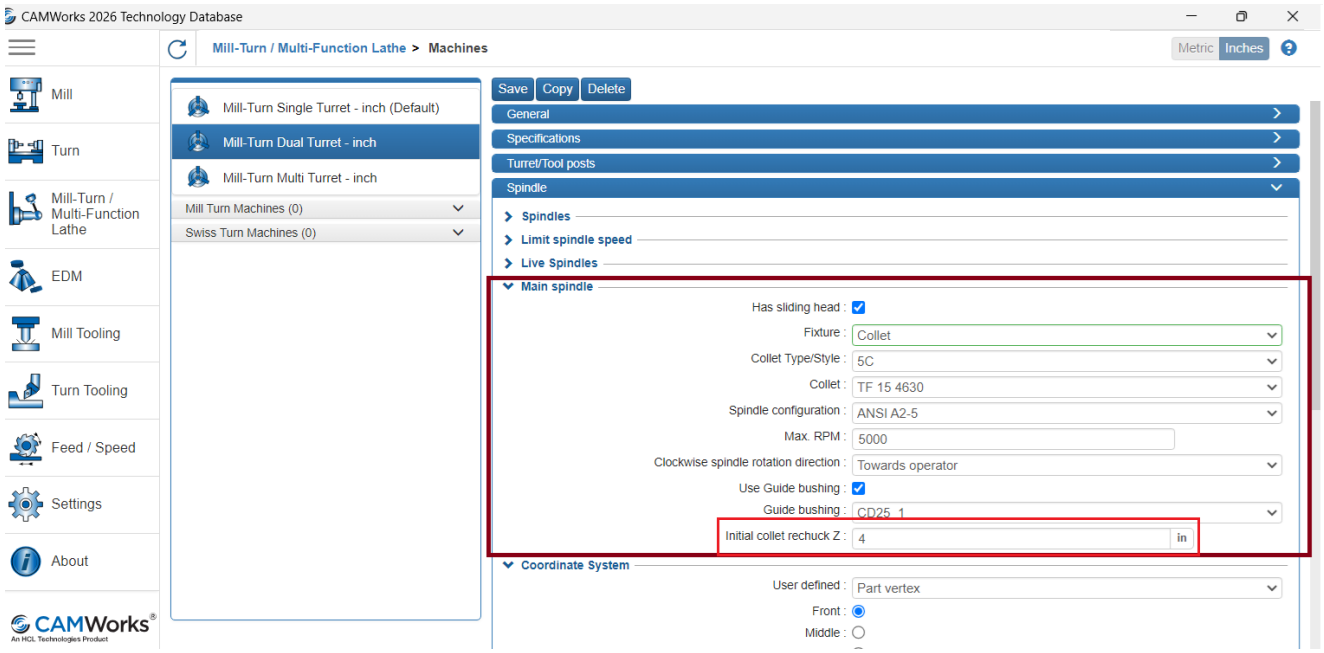


Image Illustrating indicating the 'Initial collet rechuck Z' distance when Machining a Part

Default Value of 'Initial collet rechuck Z'

The default value for the Initial collet rechuck Z parameter can be assigned to the ***Initial collet rechuck Z*** parameter under ***Spindle*** tab of the selected *Mill-Turn/Multi-Function Lathe>Machines* form within the Technology Database.



Initial collet rechuck Z parameter under Spindle Tab of Mill-Turn/Multi-Function Lathe>Machines Form of TechDB

Support Fixtures In Z Positive Space For Swiss Turn Lathes

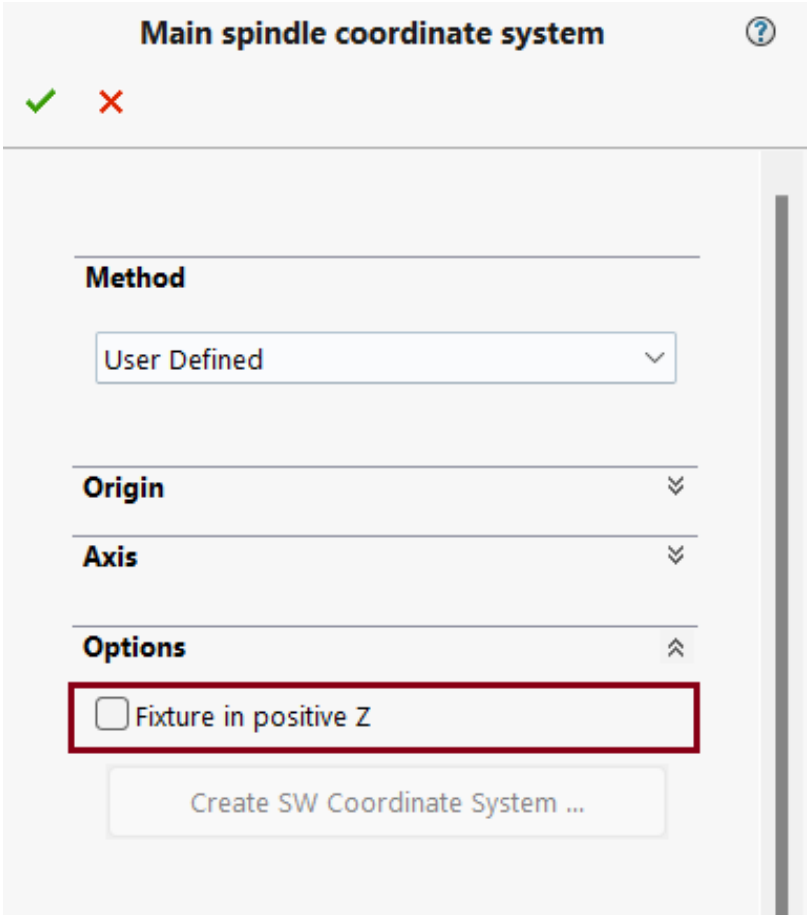
Purpose:

To allow the user to define the fixture of the main spindle in positive space of Z axis. Also, allow the coordinates of the toolpath to change as per the Z axis direction during step through and simulation.

Implementation:

In previous versions of CAMWorks, the Z axis direction for Turn, Mill-Turn machines is set such that the Main Spindle is in the Z negative direction. The direction of the Main Spindle is critical as the posted coordinates for the toolpath during Step Through, Simulation and generation of NC code is determined by the Coordinate System. However, in the Swiss Turn machines, the positive space of Z axis will be towards the main spindle.

From *CAMWorks 2026* onwards, the option to set the main spindle in Z positive direction is introduced. This option is available to you in the form of a checkbox option. The checkbox option named *Fixture in positive Z* is added under the *Main Spindle Coordinate System* dialog box.

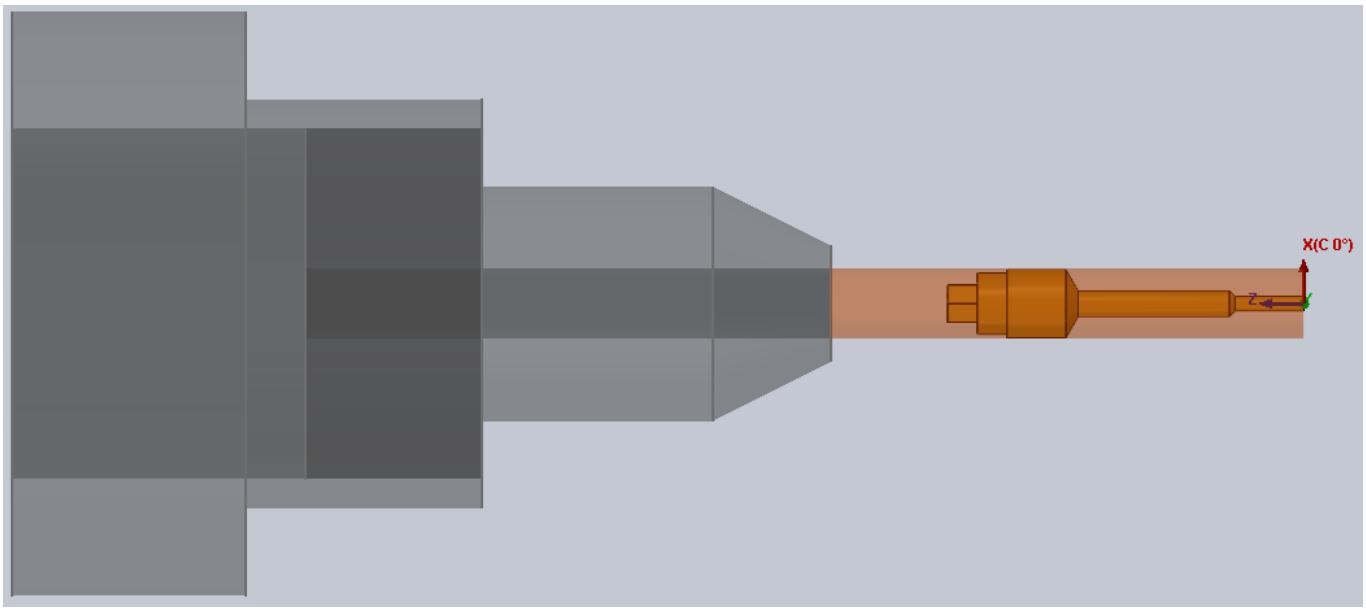


'Fixture in positive Z' checkbox option under the Main Spindle Coordinate System Dialog Box

How the Checkbox option will work:

- If the checkbox is unchecked, the Main Spindle will be in Z negative direction. (This default setting is defined in the Technology Database for the current machine.)
- If the checkbox option is checked, the fixture of the Main Spindle will be in Z positive direction.

Depending on whether the checkbox option is checked or unchecked, the posted coordinates for the toolpath during Step Through, Simulation, and in the NC code is determined and will be computed accordingly.



Main Spindle Fixture in Positive Z Direction after the 'Fixture in Positive Z' Checkbox Option is Checked

Option to Enable/Disable the Creation of Duplicate Station Numbers in Tool Cribs

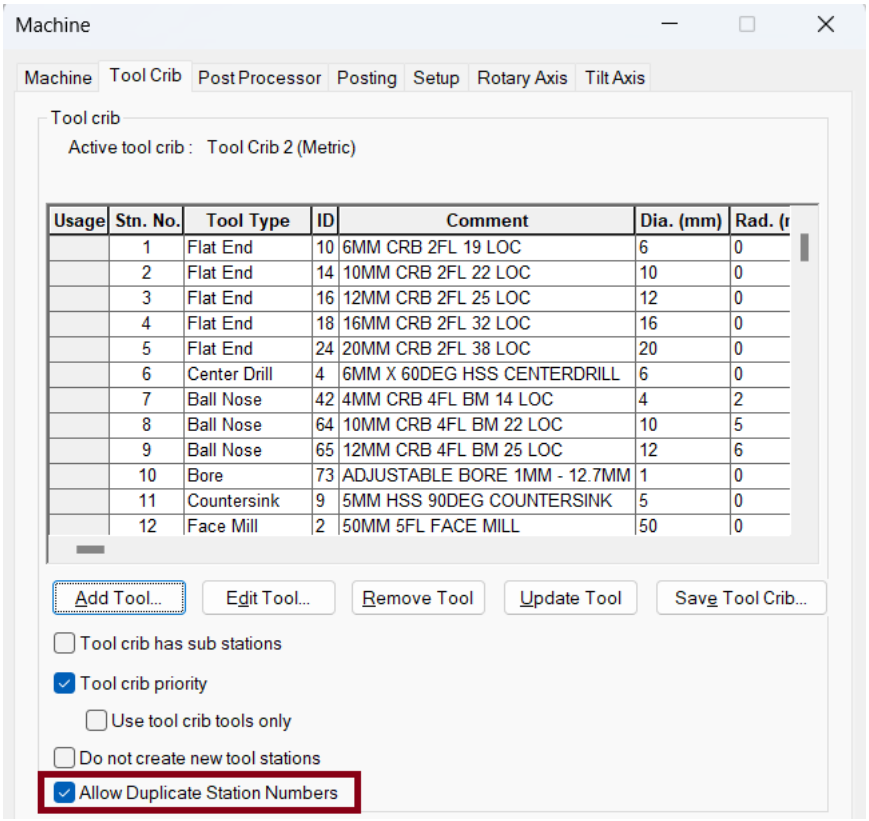
Purpose:

To provide an option to enable/disable the creation of duplicate station numbers in tool crib(s) associated with the selected machine

Implementation:

In previous versions of CAMWorks, the creation of duplicate station numbers for stations in a tool crib was allowed, but there was no option to disable this behavior. If a user wanted to prevent the creation of duplicate station numbers, no option was available to enforce it.

From CAMWorks 2026 version onwards, to address this limitation, a new checkbox option labeled ***Allow Duplicate Station Numbers*** has been introduced under the ***Tool Crib*** tab of the ***Machine*** dialog box in the CAMWorks application.



'Allow Duplicate Station Numbers' checkbox under Tool Crib tab of Machines Dialog Box

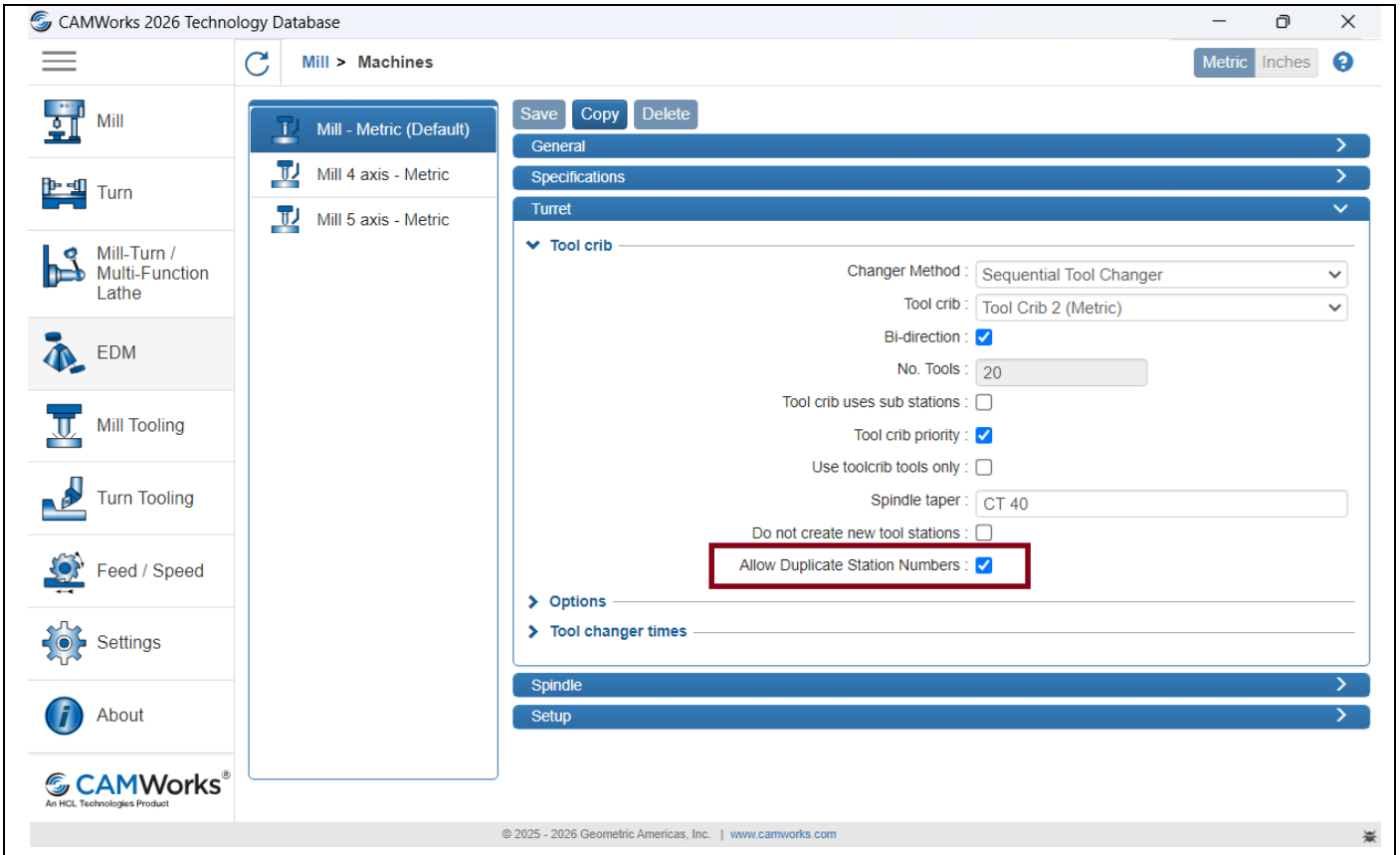
The *Allow Duplicate Station Numbers* checkbox enables users to control whether duplicate station numbers can be created in the tool cribs of the active part or assembly.

- When this checkbox is checked, CAMWorks allows stations with duplicate station numbers, including substations in the tool cribs.
- When this checkbox is unchecked, CAMWorks prevents the creation of duplicate station numbers for stations in the tool cribs.

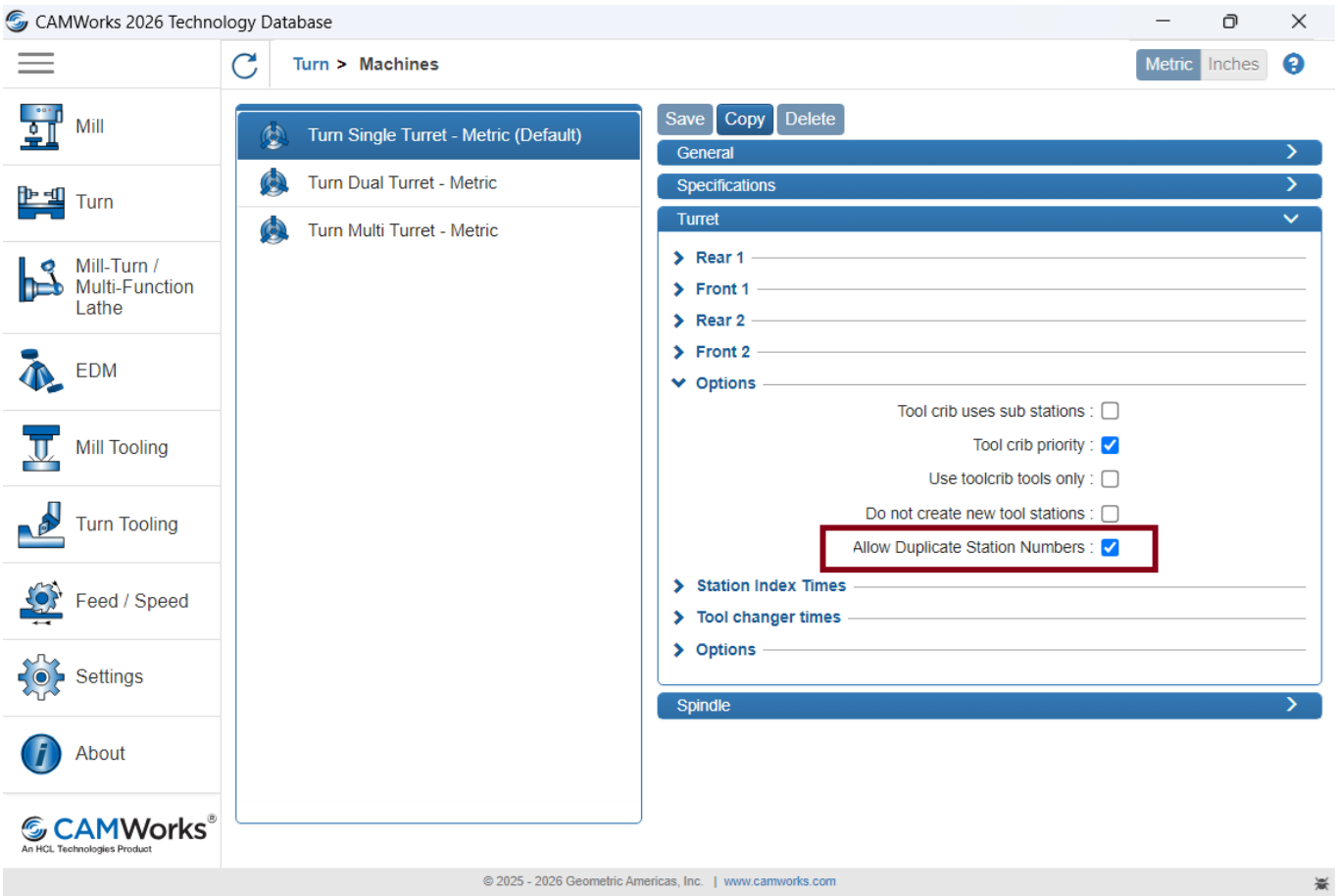
The default setting for this checkbox will depend on the setting assigned to the corresponding checkbox within the following TechDB forms:

- Under the ***Turret*** tab of the ***Machines*** form under ***Mill*** menu of TechDB
- Under the ***Turret*** tab of the ***Machines*** form under ***Turn*** menu of TechDB
- Under the ***Turret/Tool Posts*** tab of the ***Machines*** form under ***Mill-Turn/Multi-Function Lathe*** menu of TechDB

Note: For legacy parts (i.e., parts/assemblies programmed using CAMWorks 2025 or previous versions), the *Allow Duplicate Station Numbers* checkbox option will be in checked state by default.



'Allow Duplicate Station Numbers' checkbox in Turret tab of Mill > Machines Form in TechDB



'Allow Duplicate Station Numbers' checkbox in Turret tab of Turn > Machines Form in TechDB

Import and Export Machine Data in TechDB

Purpose:

To enable users to easily transfer machine configurations between different Technology Databases

Implementation:

In previous versions of CAMWorks, sharing/transferring only the machine data saved in the Technology Database (TechDB) was not possible and the entire TechDB had to be shared.

From *CAMWorks 2026* version onwards, a new functionality that allows users to export one or more machines along with all their associated data—including tool cribs, tools, and default strategies—into a single file has been introduced. This file can then be imported into another TechDB, seamlessly appending the new machine and its related data without disturbing or overwriting existing information. This functionality simplifies the process of setting up new workstations, standardizing machine definitions across an organization, and sharing configurations with other users.

The import and export capabilities are accessible via the new command buttons in the TechDB's Machine interface for the following machines:

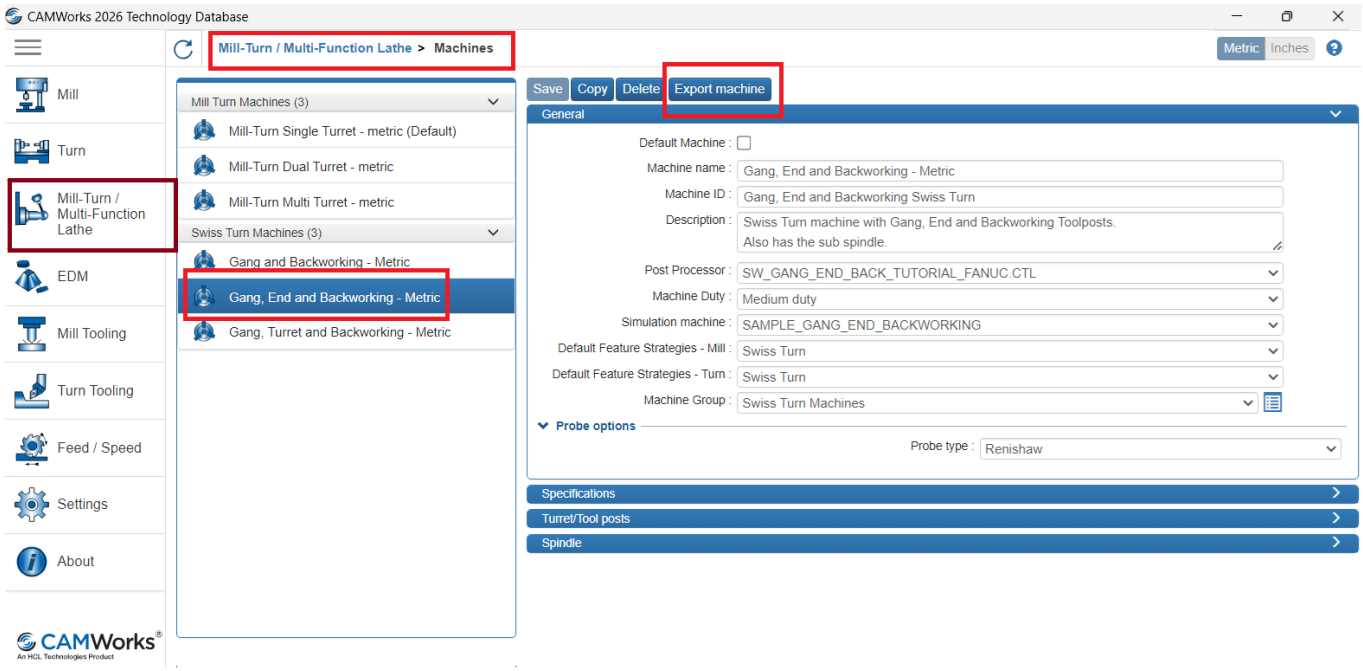
- Mill Machines
- Turn Machines (Including Dual Turret and Multi-Turret machines)
- Mill-Turn/Multi-Function Lathe Lathe/Swiss Turn Machines

Exporting a Machine

A new **Export Machine** button has been added to the TechDB's *Machine* interface for Mill, Turn and Mill-Turn machines.

To export a machine configuration:

1. Select the machine you wish to export from the list.
2. Click the **Export Machine** button.
3. A dialog box will appear, allowing you to browse to a location and assign a name to the file.
4. Upon saving, a single file with **.cwmc* file extension will be created. This file can be imported into another TechDB.



'Export Machine' button in Machines Interface for Mill-Turn/Multi-Function Lathe Machines

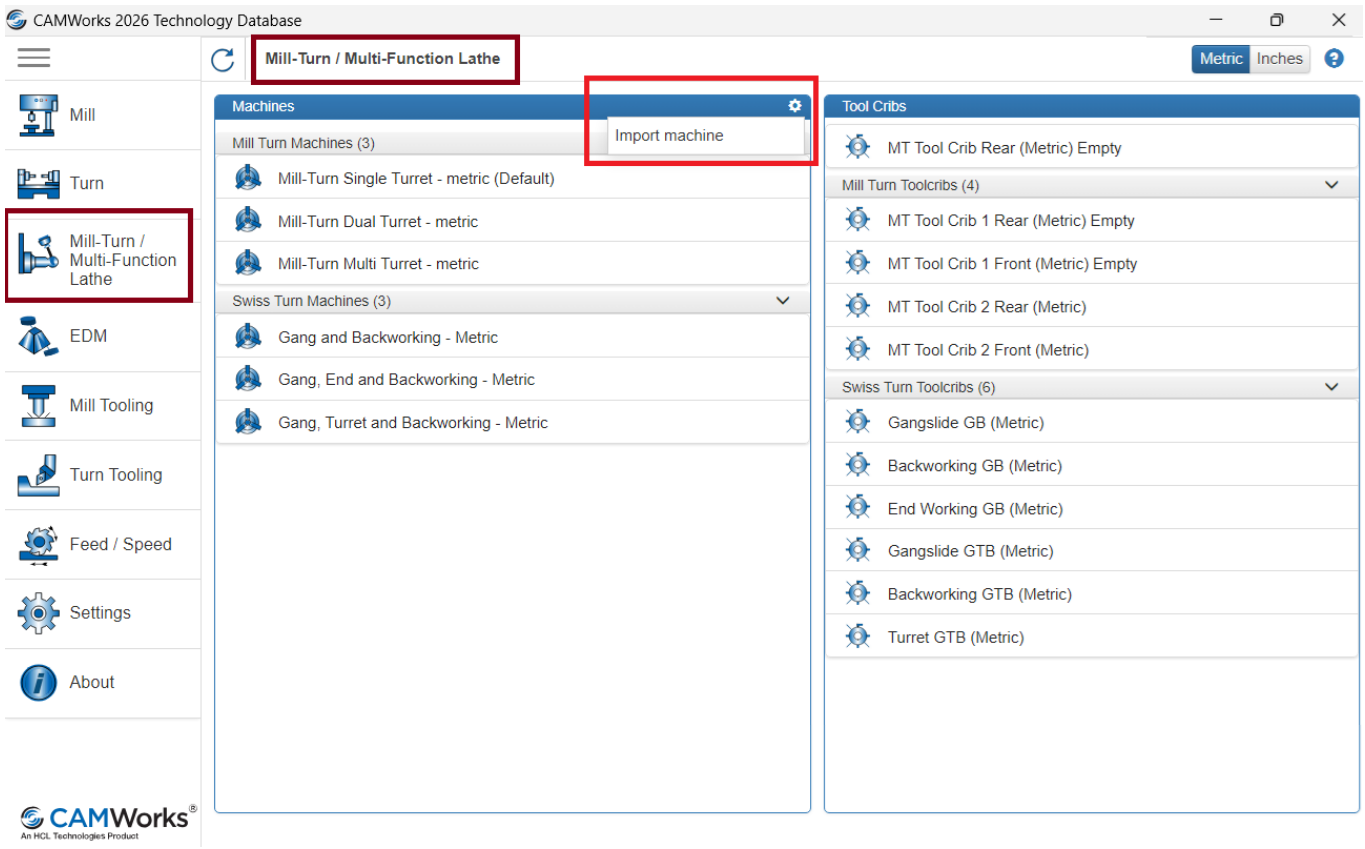
Importing a Machine

An **Import Machine** button is now available TechDB's Machine interface for Mill, Turn and Mill-Turn/Multi-Function Lathe machines.

To import a machine configuration:

1. Click the ***Import Machine*** button .
2. Browse and select the ****.cwmc*** file you wish to import.
3. Upon import, the system will:
   i. Create a new machine in the active TechDB with the corresponding unit type.
   ii. Append all the associated tool cribs and tools from the file to the existing database.
   iii. Set the post processor and default strategies based on the available options in the target TechDB.

The newly imported machine and its data can be edited just like any other entry in your TechDB.



'Import Machine' button in Machines Interface for Mill-Turn/Multi-Function Lathe Machines

Indicating whether a Tool Post/ Turret/ Gang Slide of Swiss-Turn Machines is capable of movement in Z axis

Purpose:

Option to indicate whether a Tool Post/ Turret/ Gang Slide is capable of movement in Z axis or not

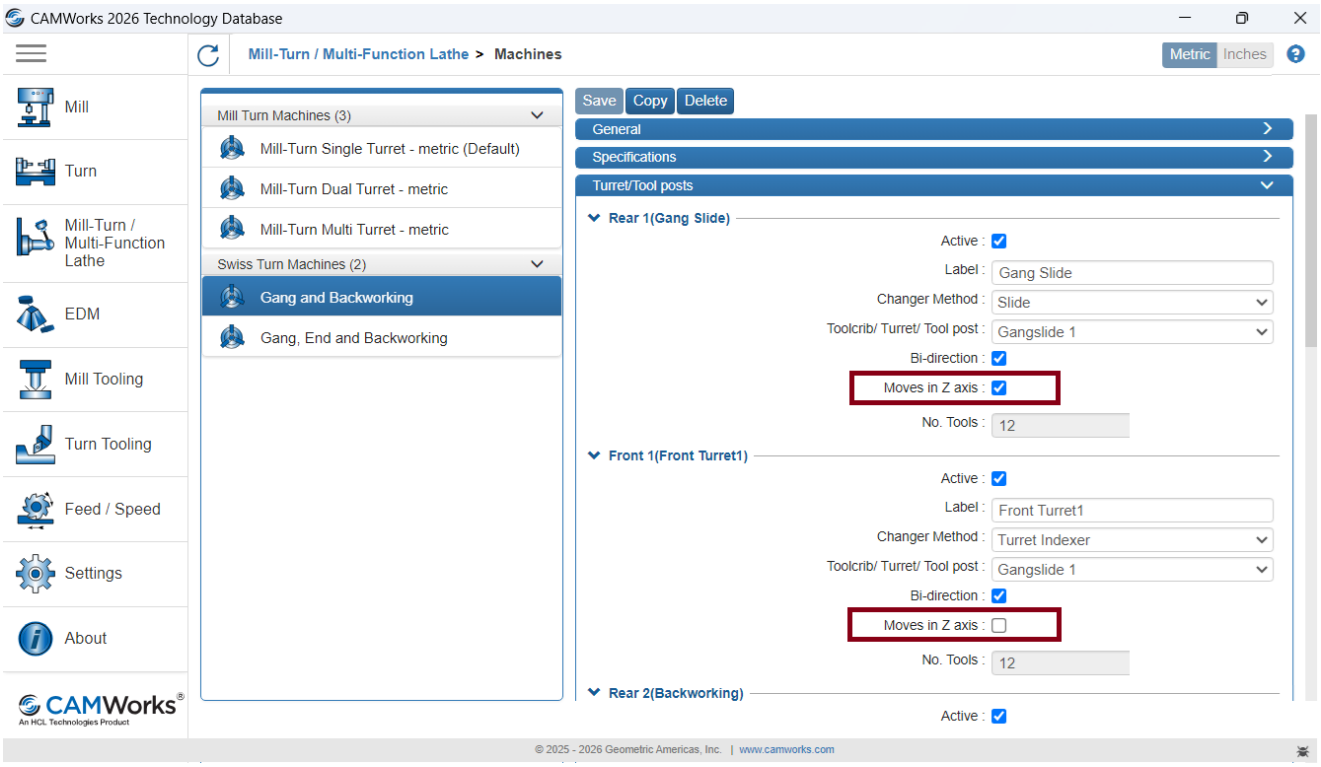
Implementation:

For a Mill-Turn/Multi-Function Lathe machine with Sliding Head, some of the tool posts can move only in X axis or Y axis direction. Such Tool Posts/ Turrets/ Gang Slides do not have movement in Z axis direction. It is necessary to indicate whether the tool post or stock will provide the Z axis movement for the toolpath to ensure error free machining.

From *CAMWorks 2026* version onwards, a checkbox option labelled *Moves in Z Axis* has been introduced. In the TechDB, this option is available for all Turret/Tool posts under the Turret/Tool posts tab of the Mill-Turn/Multi-Function Lathe > Machines form. This checkbox is unchecked by default.

Use this checkbox option to specify if the Turret/Tool Post has movement in Z Axis or not.

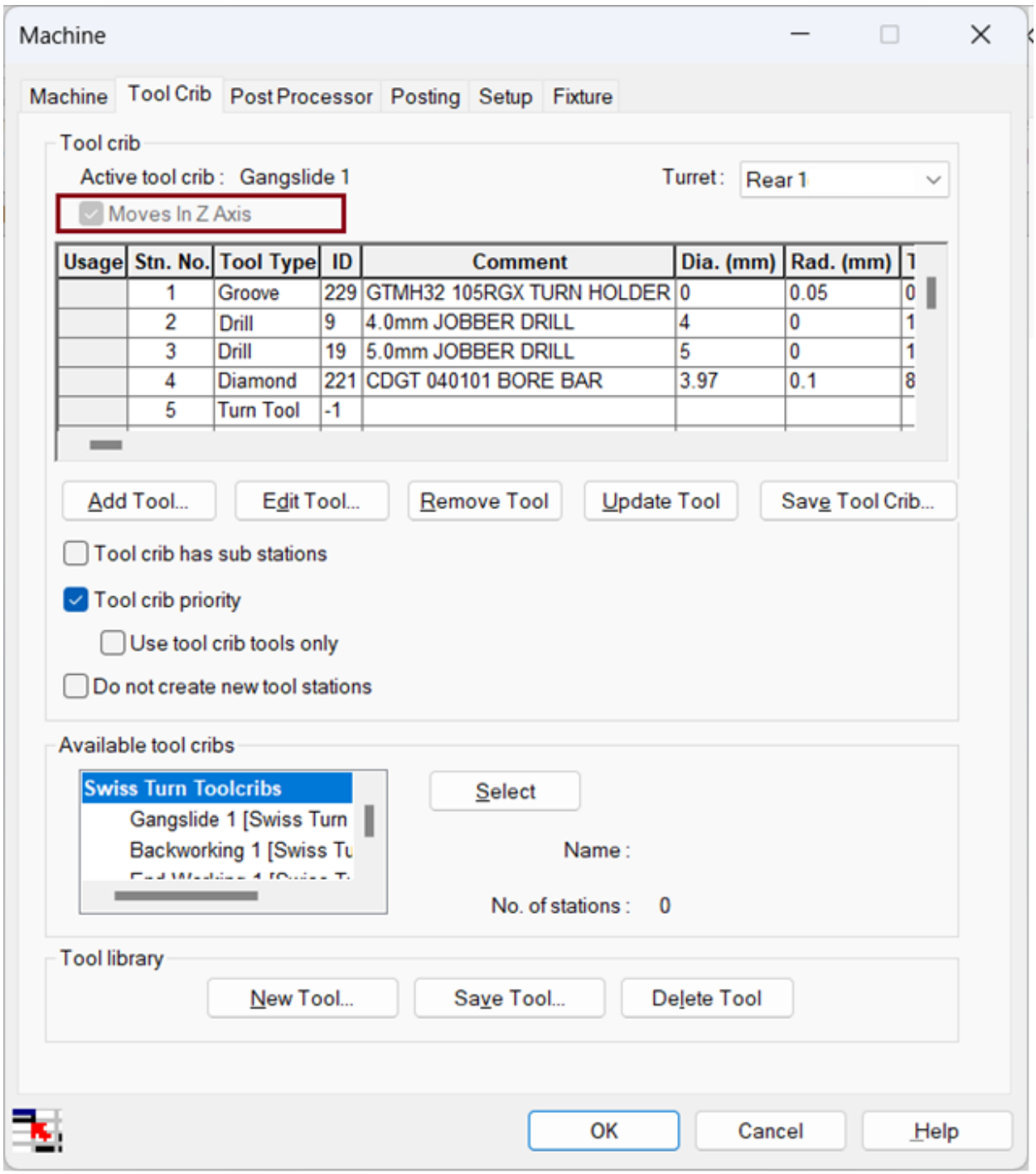
- If this checkbox option is unchecked, it indicates that the selected Turret/Tool Post has no movement in Z axis. The Z axis movement when machining the toolpath will be provided by the stock.
- If this checkbox option is checked, it indicates that the selected Turret/Tool Post has movement in Z axis. The Z axis movement when machining the toolpath will be provided by the tool assigned to that operation. (The tool is loaded on the selected Turret/Tool Post.)



'Moves in Z axis' checkbox option under the Turret/Tool posts tab of the Mill-Turn/Multi-Function Lathe > Machines form

Within the *CAMWorks* application, the *Moves in Z Axis* checkbox option will be displayed under the Tool Crib tab of Machine dialog box as a read-only field.

This checkbox will be displayed only if the active machine is a Mill-Turn/Multi-Function Lathe machine with a sliding head (i.e., the *Has Sliding Head* checkbox under Fixture tab of the Machine dialog box is checked). Its checked/unchecked status will be determined by the selection made for corresponding *Moves in Z Axis* checkbox under Turret/Tool Posts tab under the *Mill-Turn/Multi-Function Lathe>Machines* form within the TechDB.



'Moves in Z axis' checkbox option under the Tool Crib tab of Machine dialog box as a read-only field

User-assigned names for Turrets / Tool posts associated with Mill-Turn / Swiss Lathes / Multi-Function Lathes

Purpose:

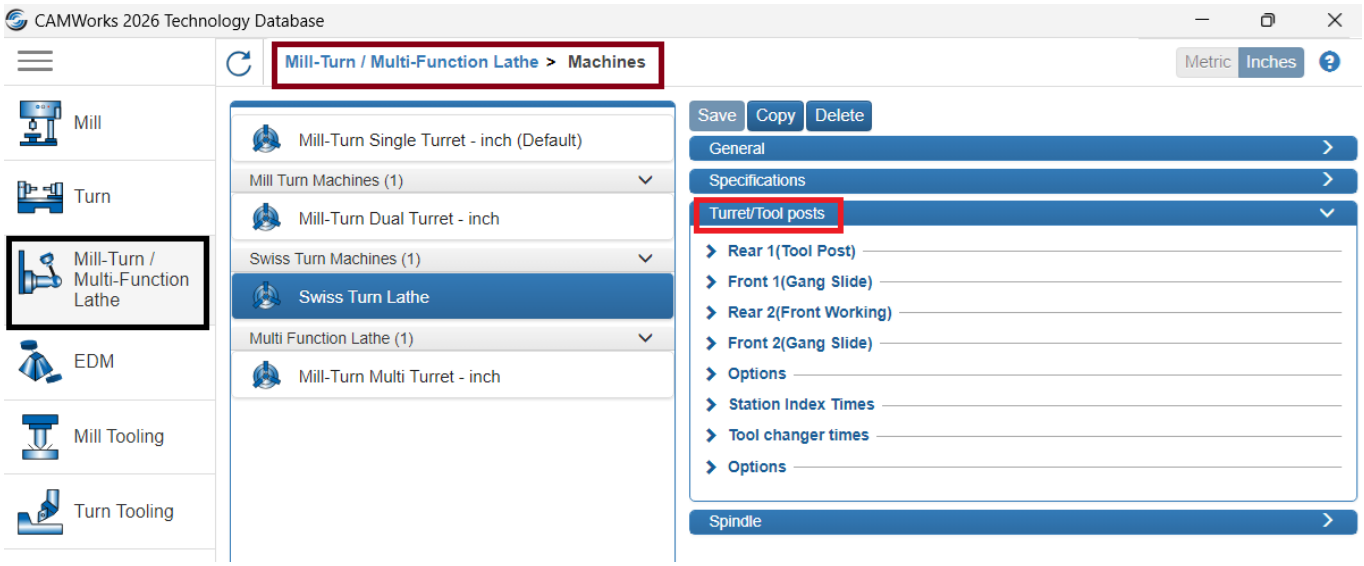
To provide the option to assign user-assigned names for Turrets / Tool posts associated with Mill-Turn/ Swiss Lathes/ Multi-Function Lathes so that they indicate the turret / tool post type (Gang Slide, Tool Post, etc.)

Implementation:

In the Technology Database, the ***Machines*** form present in Mill, Turn and Mill-Turn/Multi-Function Lathe menus is used for viewing, creating, editing, and/or deleting machine definitions. The ***Turret /Tool posts*** tab within the Machines form is used for viewing, defining, and editing information associated with its tool cribs (usually turrets).

Renaming of Turret tab to Turret/Tool Posts Tab

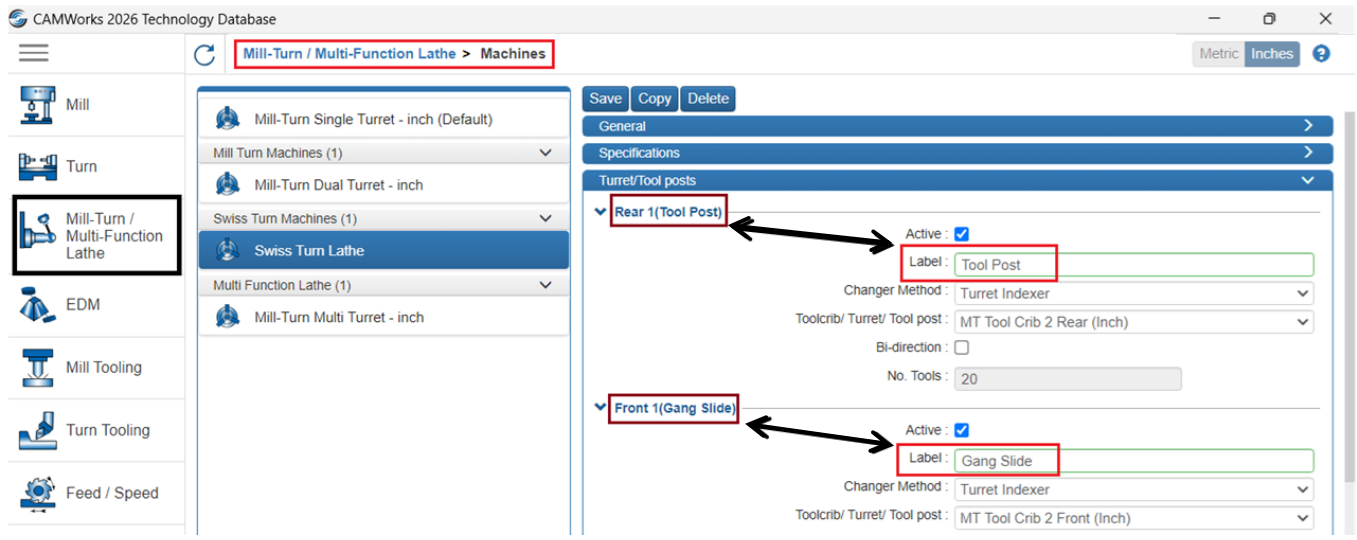
From *CAMWorks 2026* version onwards, within the ***Mill-Turn/Multi-Function Lathe> Machines*** form, the ***Turret*** tab has been renamed to ***Turret/Tool Posts*** tab.
The renaming is deliberate as this specific tab can now be used for viewing, creating, or editing turret / tool posts definitions for Swiss Lathe machines in addition to regular CNC Lathe machines. While regular CNC Lathe/Mill-Turn machines have term of ***Turret***, the Swiss Lathes have different names viz. ***Tool Posts***, ***Turrets***, ***Gang Slides***, ***End Working*** etc. The updated label of the tab serves as a visual indicator of the expanded scope of this tab.



Turret/Tool Posts Tab under Mill-Turn/Multi-Function Lathe Machines Form within TechDB

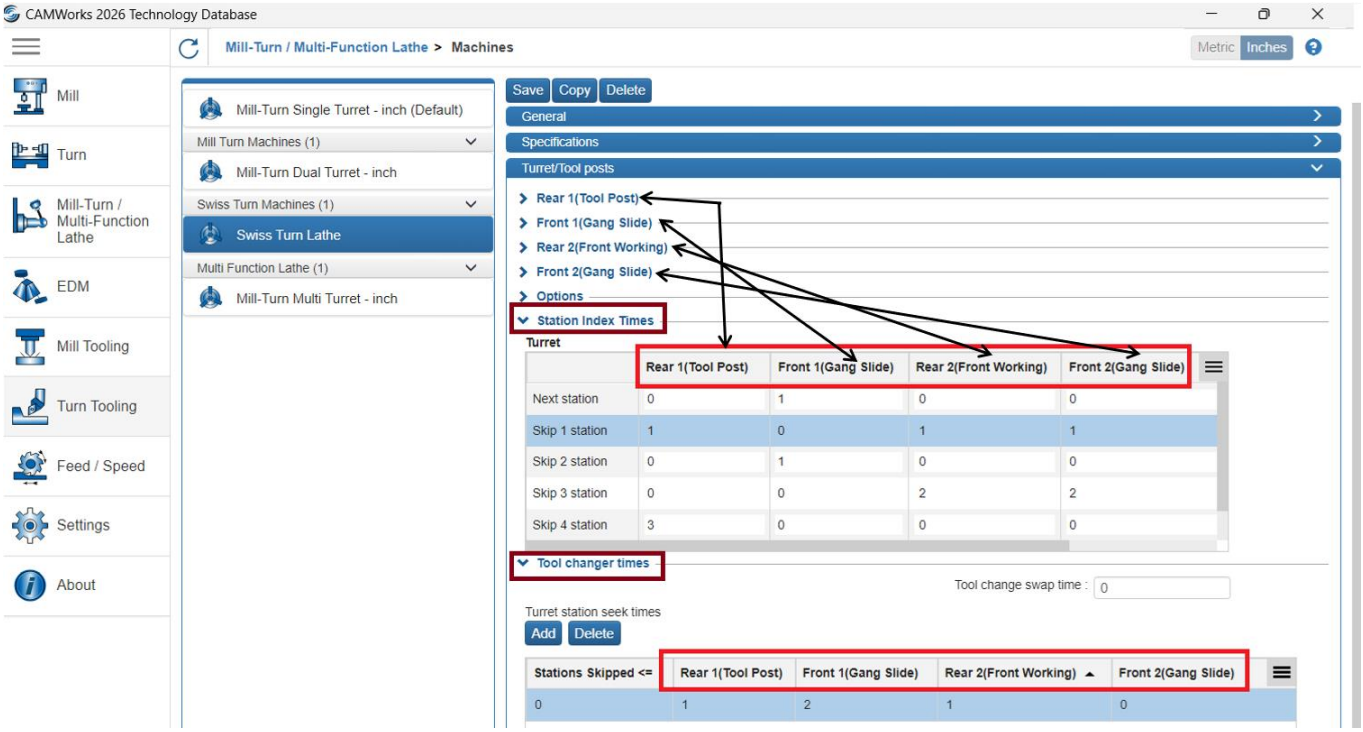
Label Parameter under Turret/Tool Posts Tab

The first four collapsible group boxes under the *Turret/Tool Posts* tab are to define the number of tool posts for the selected machine along with their associated properties. Each of these group boxes represents a tool post that can be activated/deactivated as per the requirements for the specific machine definition. In each of these group boxes, a parameter named ***Label*** is present. Use this parameter to provide a short label for the specific turret / tool post.
It is highly recommended that you assign a label that provides information about the tool posts function/order/location/type. This is necessary as the label you assign will be displayed in the label for the specific Tool Post. Observe the title header of the Turret / Tool posts group box (*Rear 1*, *Front 1*, *Rear 2* and *Front 2*). The label you assign will be dynamically updated and displayed within parentheses as a suffix to the title header of the *Turret/Tool posts* group boxes.



The Label assigned within the Tool Crib Group box will serve as the Name/Label for that Tool Crib

The labels you assign to the Turret will also be displayed as column headers for the tables displayed under group boxes labelled ***Station Index Times*** and ***Tool Changer Times*** under the *Turret/ Tool Posts* tab of the *Mill-Turn/Multi-Function Lathe> Machines* form.



Tables under 'Station Index Times' and 'Tool Changer Times' group boxes displaying User-Assigned Turret / Tool post Names as Column Headers

Options to Display Default or Custom Names for Turrets/Tool Posts of Mill-Turn/Swiss Turn Machines

Purpose:

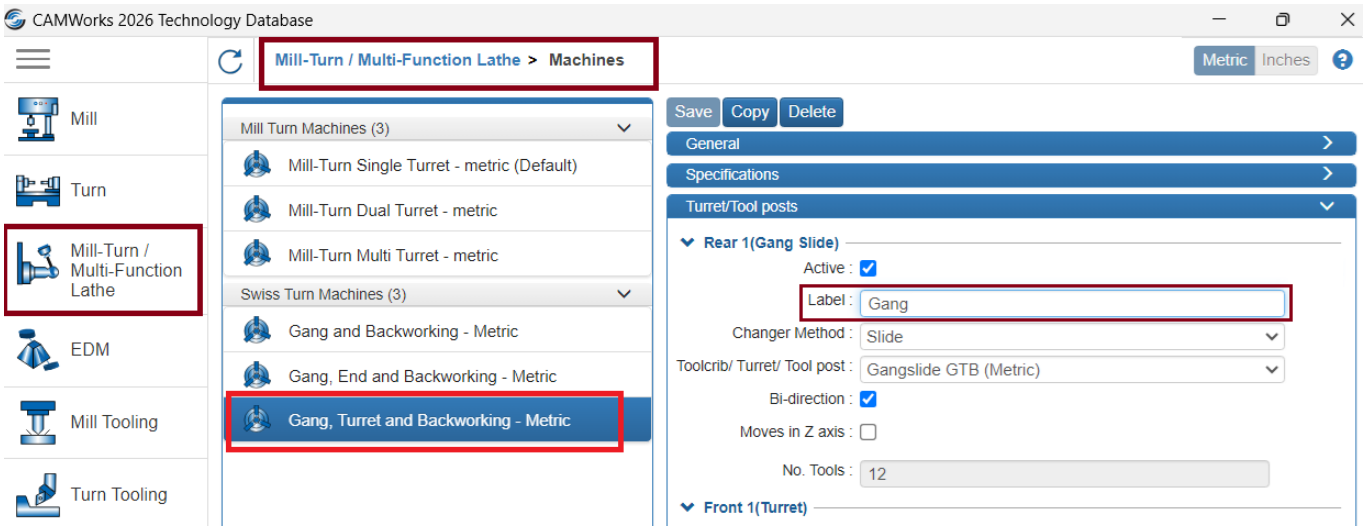
To provide the option to display either their default names or labels (customized names) for Mill-Turn and Swiss-Turn Turrets/ Tool Posts nodes displayed in the Tools Tree tab

Implementation:

The nomenclature for *Mill-Turn* and *Swiss-Turn* Turrets/Tool Posts nodes displayed in the ***Tools Tree*** tab is:**<Name of Tool Crib loaded in the Turret> [Turret Name]**.

For example, if the name of the tool crib loaded in '***Rear1***' turret is **'MT Tool Crib Rear (Metric)'**, then the name of the turret node displayed in the Tools tree will be **'MT Tool Crib Rear (Metric) [Rear1]'**.

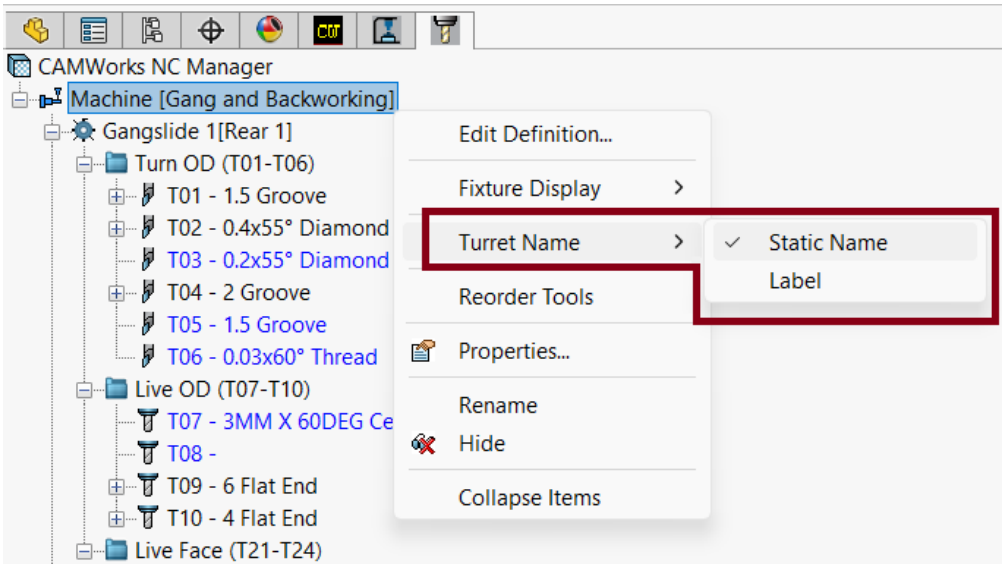
For Mill-Turn and Swiss-Turn Turrets/ Tool Posts nodes displayed in the *Tools Tree* tab, CAMWorks provides the option to display either their default names or labels (customized names) within the square brackets. The custom name is the user-defined label that you have assigned to the turrets/Tool Posts within the *Turret/Tool Posts* tab of the *Machines* form for Mill-Turn and Swiss-Turn machines of the TechDB.



'Label' field under 'Turret/Tool Posts' Tab of Mill-turn/Multi-Function Lathe Machines User Interface

'Turret Name' Option in Context Menu of Machine Node in Tools Tree

When you right-click on the ***Machine*** node in the *Tools Tree* tab, this option will be available as ***Turret Name*** item within the context menu.



'Turret Name' Option in Context Menu of Machine Node in Tools Tree

When you mouse hover over the ***Turret Name*** menu item of the context menu, the following two options will be available:

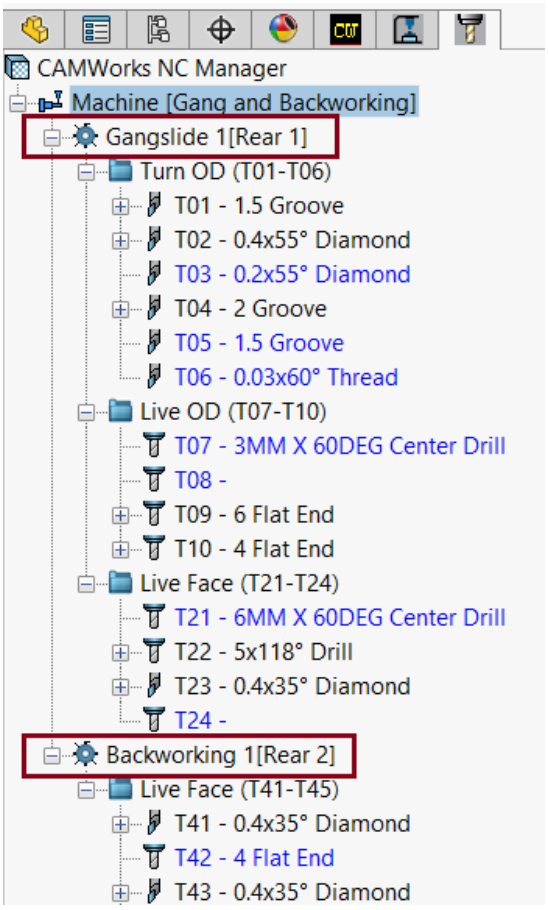
- **Static Name**

  In CAMWorks, the syntax of the default names that are assigned to rear turrets and front turrets are 'Rear #' and 'Front #' respectively. ('#' within this syntax indicates the incremental serial number starting from the number 1. For example, Rear 1, Rear 2, Front 1, Front 2, etc. These names are static in nature and cannot be changed.

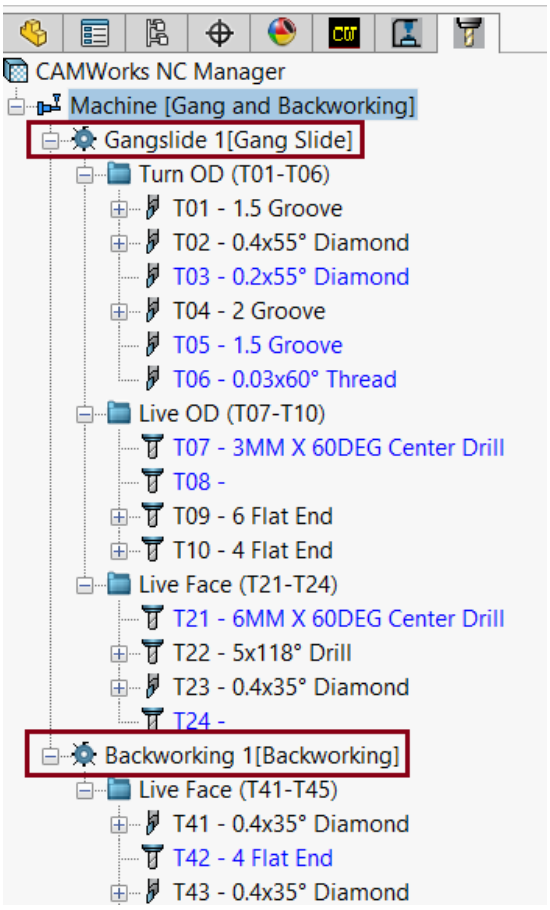
  When the ***Static*** option is selected for *Turret Name*, the default names of the turrets/Tool Posts will be displayed for their corresponding nodes in the Tools tree within square brackets.

- **Label**

  When this option is selected, the user-defined label that you have assigned to the turrets/Tool Posts within the *Turret/Tool Posts* tab of the *Machines* form for Mill-Turn and Swiss-Turn machines of the TechDB will be displayed as the names for the turrets/Tool Posts nodes in the *Tools Tree* within square brackets.



Static Name Displayed in Square Brackets for Turret/Tool Posts Nodes



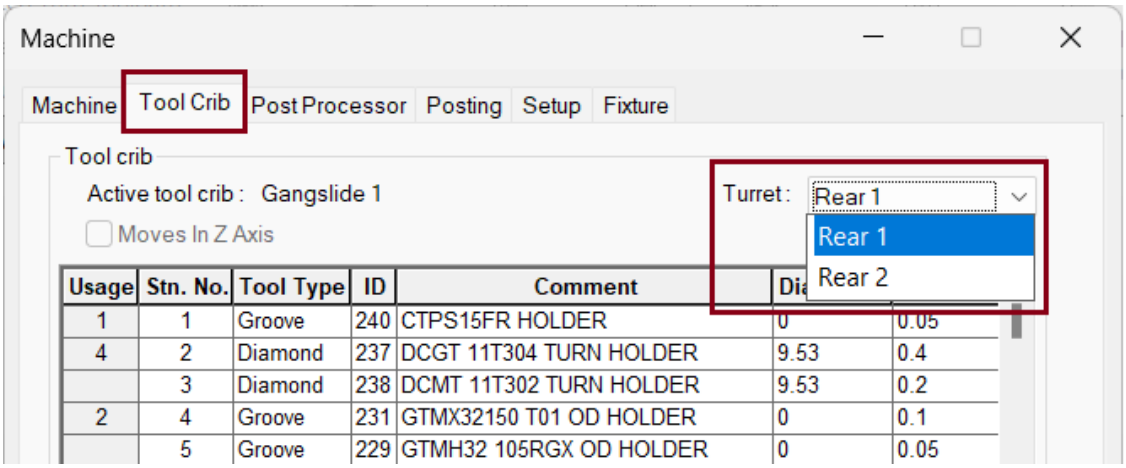
Labels Displayed in Square Brackets Tools for Turret/Tool Posts Nodes

How 'Turret Name' Setting Affects Other User Interfaces of the CAMWorks Application

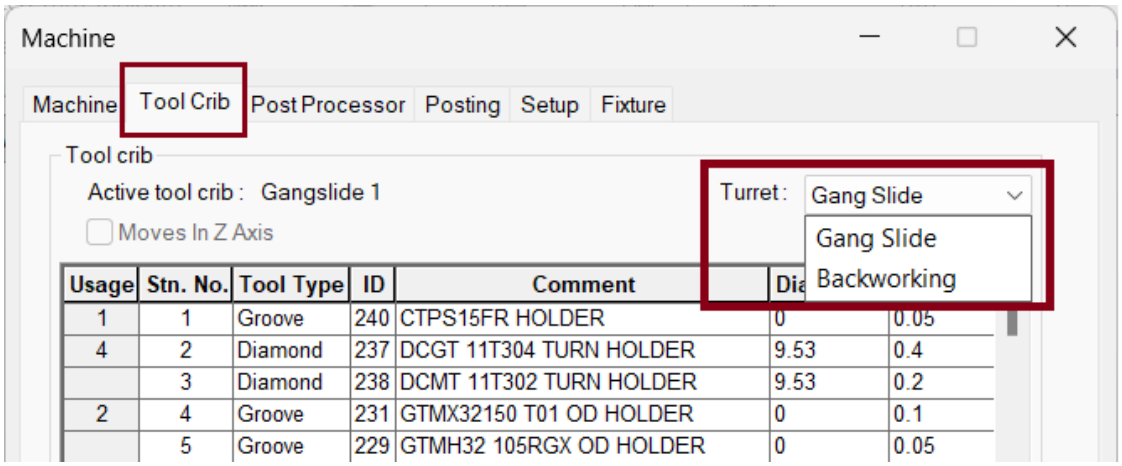
For *Mill-Turn/Swiss-Turn* machines, the setting (***Static Name*** or ***Label***) assigned to *Turret Name* option available in the context menu of the *Machine* node in the *Tools Tree* will affect the following user interfaces within the CAMWorks application:

- **'Turret' Dropdown list under Tool Crib Tab of Machine Dialog Box**

  If the *Turret Name* option is set to ***Static Name***, then the default names of the turrets will be displayed in the dropdown list. If set to ***Label***, then the user-defined label you have assigned to the active Turrets within the *Turret/Tool Posts* tab of the *Machines* form in the TechDB will be displayed in this dropdown list.



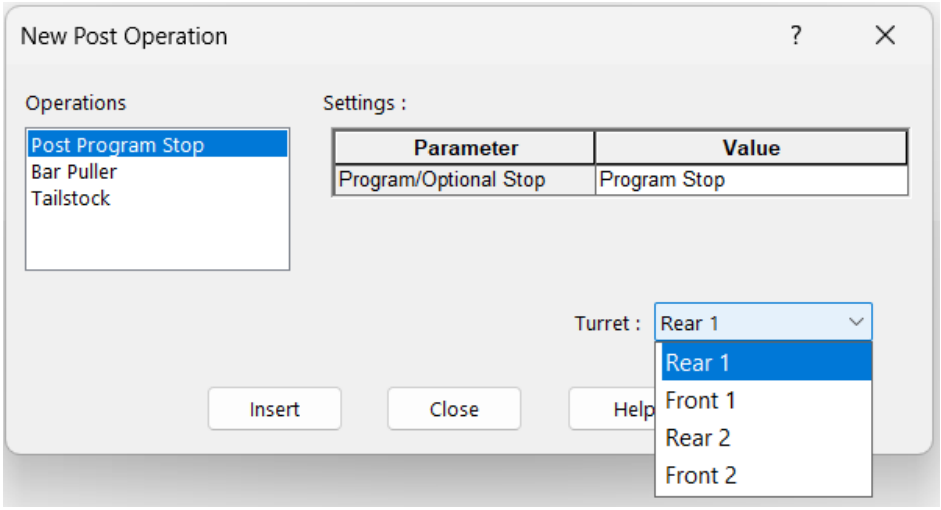
Turret Dropdown list when 'Turret Name' Option is set to 'Static Name'



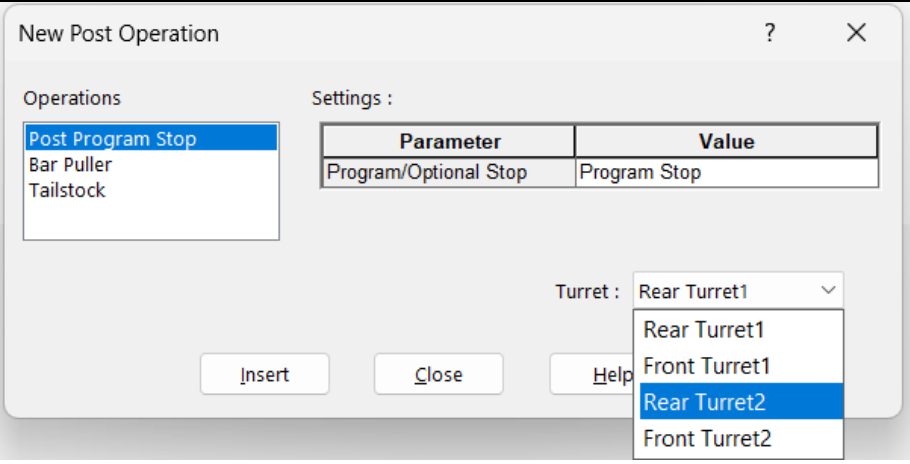
Turret Dropdown list when 'Turret Name' Option is set to 'Label'

- **Turret Dropdown list in New Post Operation Dialog Box**

  The setting assigned to the *Turret Name* option in the context menu for the *Machine* node in the Tools Tree influences the display of the turret names in *Turret* dropdown list in *New Post Operation* dialog box.



Turret Dropdown list of New Post Operation Dialog Box when 'Turret Name' Option is set to 'Static Name'

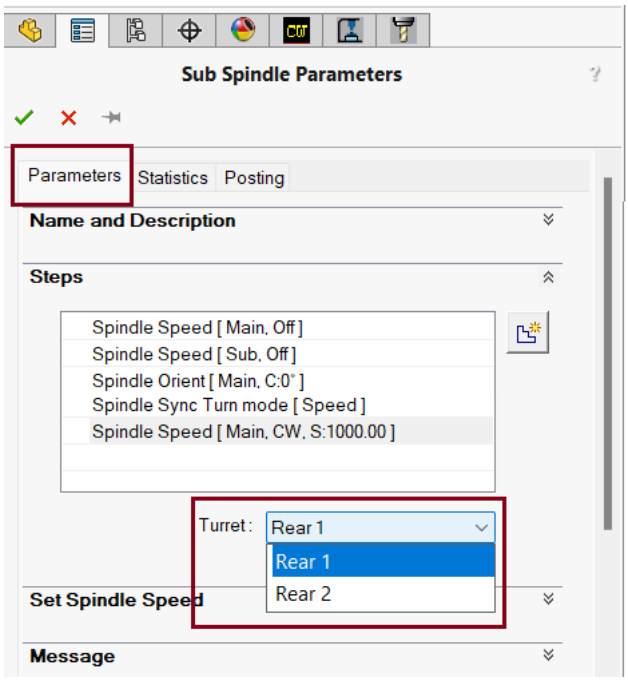


Turret Dropdown list of New Post Operation Dialog Box when 'Turret Name' Option is set to 'Label'

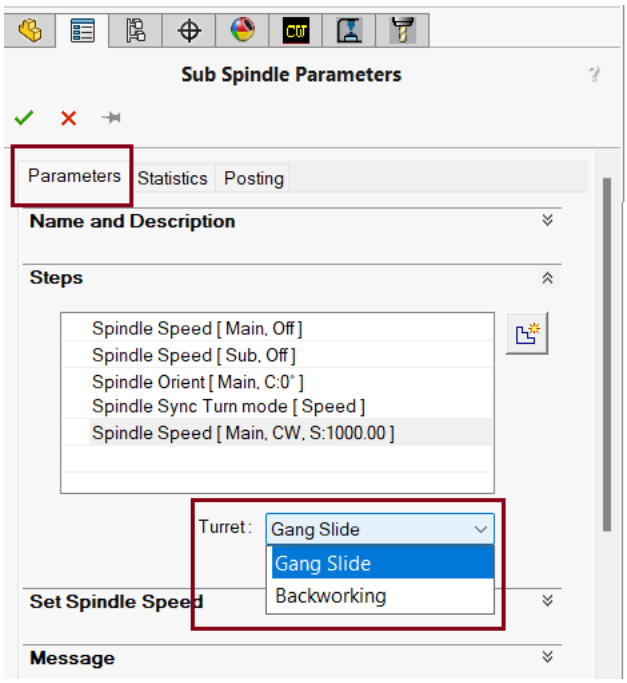
- **Turret Dropdown list under Sub Spindle Parameters: Parameters Tab**

  The setting assigned to the *Turret Name* option in the context menu for the *Machine* node in the Tools Tree influences the display of the turret names in *Turret* dropdown list under *Sub Spindle Parameters: Parameters tab* dialog box.

  - If the *Turret Name* option is set to *Static Name*, then the default names of the turrets will be displayed in the Turret dropdown list.
  - If the *Turret Name* option is set to *Label*, then the user-defined labels assigned to the turrets within the *Turret/Tool Posts* tab of the *Machines* form for Mill-Turn and Swiss-Turn machines of the TechDB will be displayed in this dropdown list.



Turret Dropdown list under Sub Spindle Parameters Dialog Box when 'Turret Name' Option is set to 'Static Name'

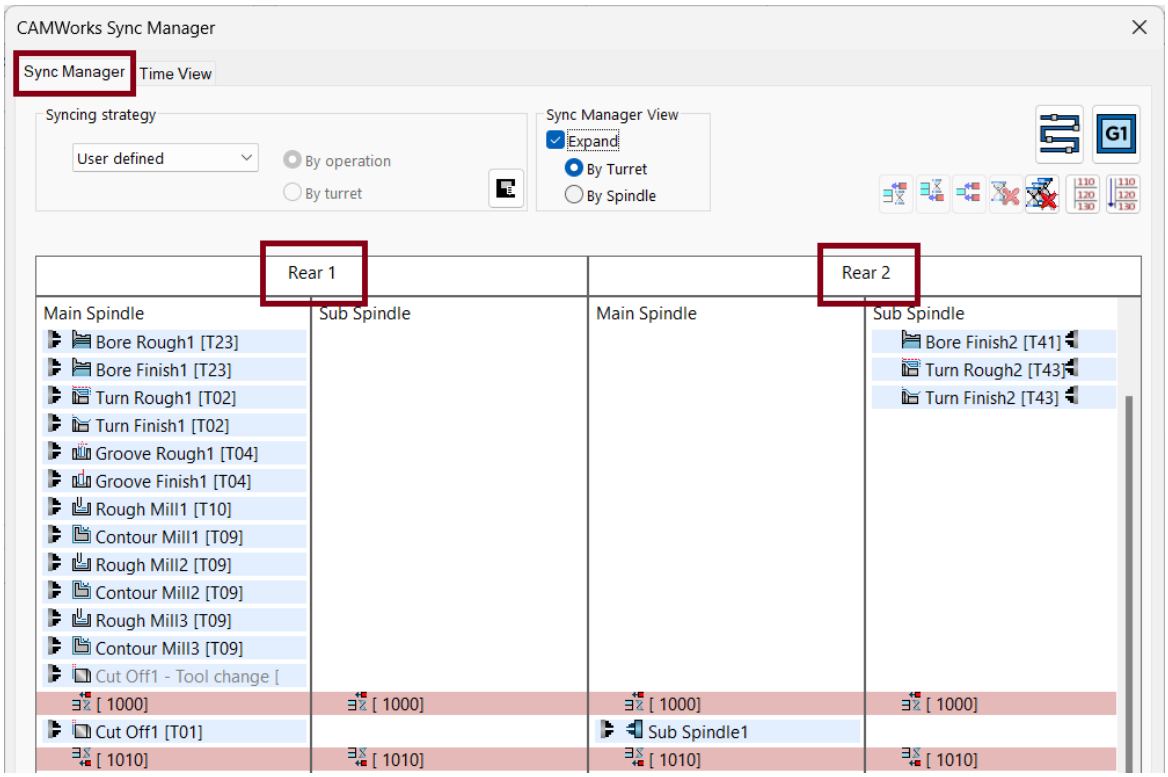


Turret Dropdown list under Sub Spindle Parameters Dialog Box when 'Turret Name' Option is set to 'Label'

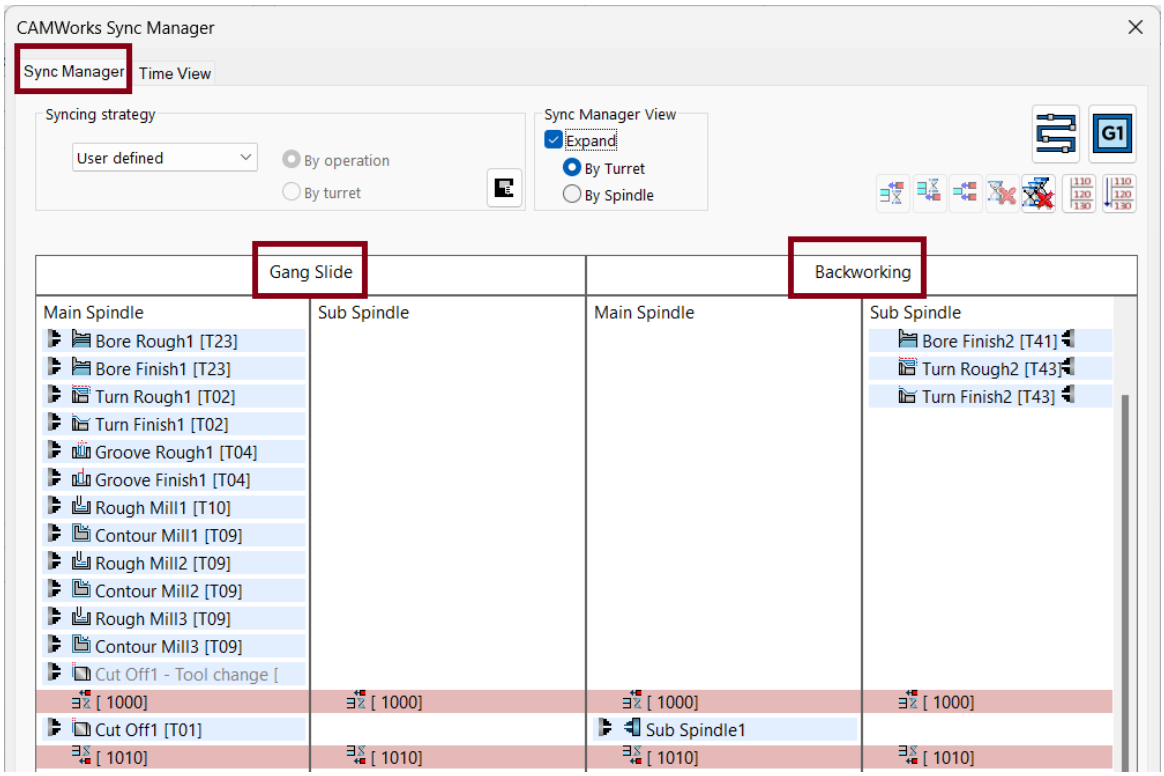
- **Turret Names Displayed in the under Sync Manager Tab of CAMWorks Sync Manager**

  The setting assigned to the *Turret Name* option in the context menu for the Machine node in the Tools Tree influences the display of the Turret dropdown list under *Tool Crib* tab of the Mill-Turn *Machine* dialog box.

- If the *Turret Name* option is set to *Static Name*, then the default names of the turrets will be displayed as column headers in the *Sync Manager Grid* within their respective columns.
- If the *Turret Name* option is set to Label, then the user-defined labels assigned to the turrets within the *Turret/Tool Posts* tab of the *Machines* form for Mill-Turn and Swiss-Turn machines of the TechDB will be displayed as column headers in the *Sync Manager Grid* within their respective columns.



Turret Column Headers under Sync Manager when 'Turret Name' Option is set to 'Static Name'

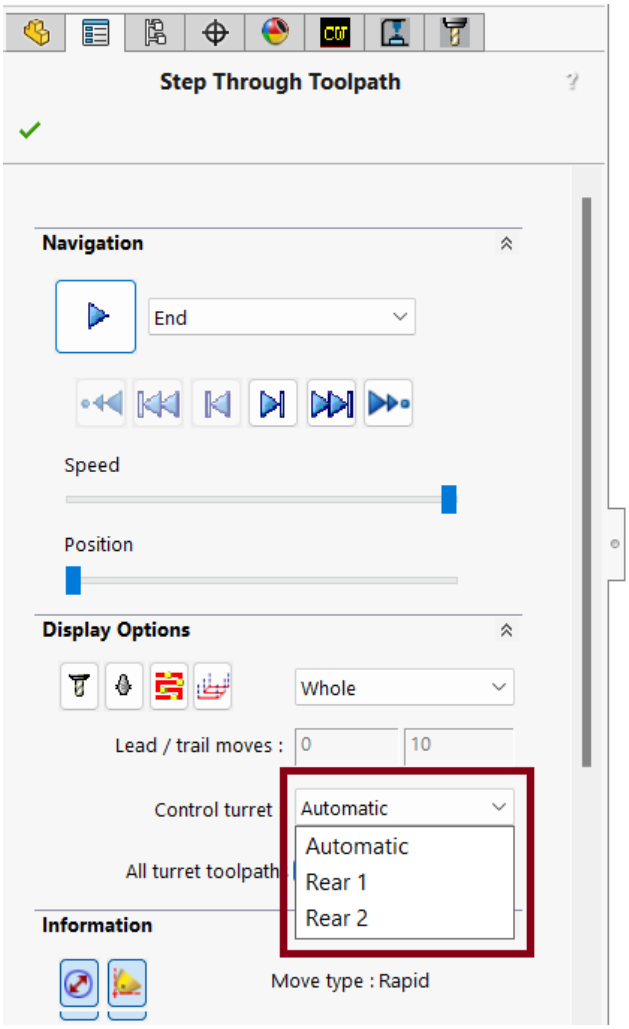


Turret Column Headers under Sync Manager when 'Turret Name' Option is set to 'Label'

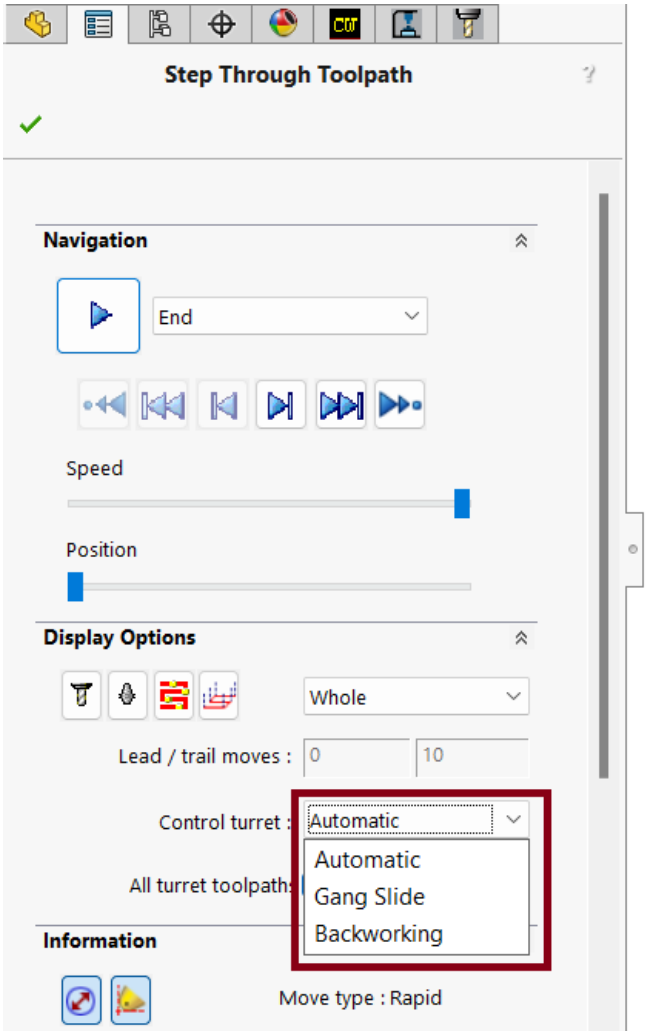
- **'Control turret' Dropdown list in Step Through Toolpath Dialog Box**

  The setting assigned to the *Turret Name* option in the context menu for the *Machine* node in the Tools Tree influences the display of the turret names in ***Control turret*** dropdown list in *Step Through Toolpath* dialog box.

  - If the *Turret Name* option is set to *Static Name*, then the default names of the turrets will be displayed in the Turret dropdown list.
  - If the *Turret Name* option is set to *Label*, then the user-defined labels assigned to the turrets within the *Turret/Tool Posts* tab of the *Machines* form for Mill-Turn and Swiss-Turn machines of the TechDB will be displayed in this dropdown list.



Control Turret Dropdown list in Step Through Toolpath Dialog Box when 'Turret Name' Option is set to 'Static Name'

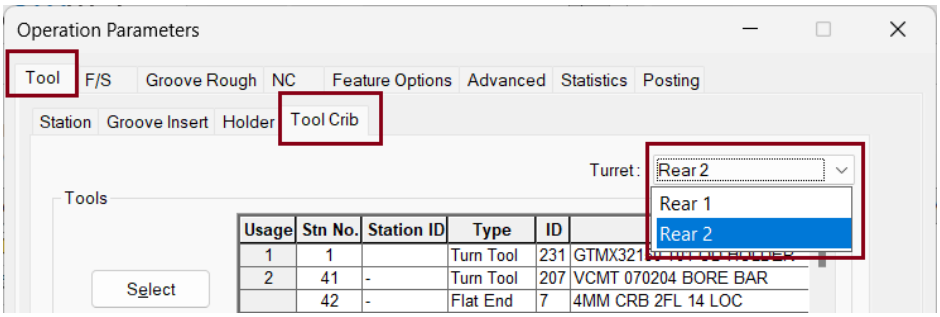


Control Turret Dropdown list in StepThrough Toolpath Dialog Box when 'Turret Name' Option is set to 'Label'

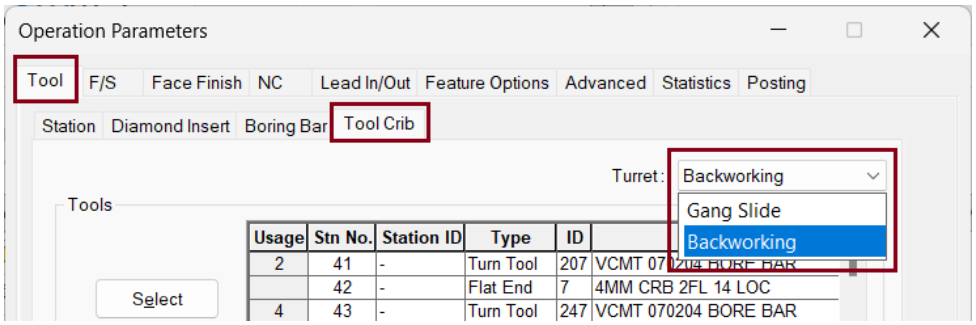
- **Turret Dropdown list under Tool Crib page under Tool Tab of Operation Parameters dialog box**

  The setting assigned to the *Turret Name* option in the context menu for the *Machine* node in the Tools Tree influences the display of the turret names in *Turret* dropdown list under *Tool Crib* page of Tool tab in Operation Parameters dialog box.

  - If the *Turret Name* option is set to *Static Name*, then the default names of the turrets will be displayed in the Turret dropdown list.
  - If the *Turret Name* option is set to *Label*, then the user-defined labels assigned to the turrets within the *Turret/Tool Posts* tab of the *Machines* form for Mill-Turn and Swiss-Turn machines of the TechDB will be displayed in this dropdown list.



Turret Dropdown list under Tool Crib Page of Tool Crib tab in Operation Parmeters Dialog Box when Turret Name is set to 'Static Name'

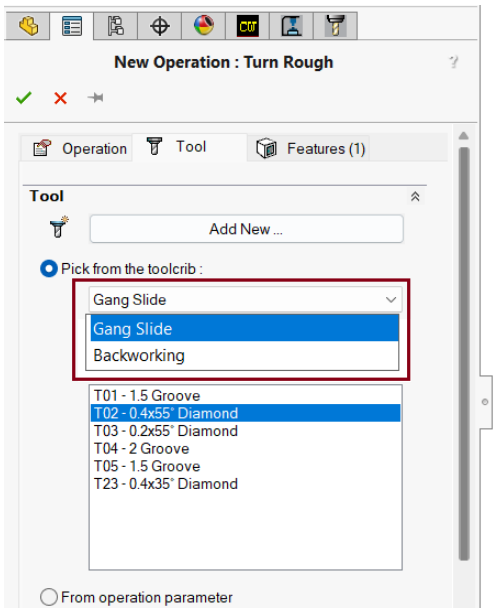


Turret Dropdown list under Tool Crib Page of Tool Crib tab in Operation Parmeters Dialog Box when Turret Name is set to 'Label'

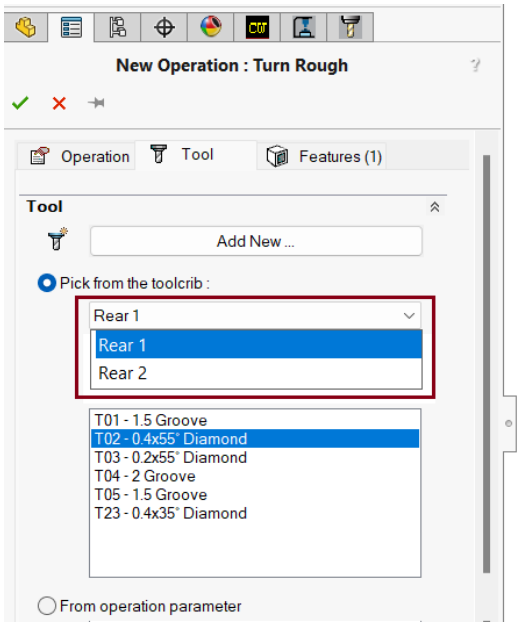
- **Turret names displayed under Tool Tab when interactively inserting new operations**

  Whenever new operations are generated using Generate Operation Plan command or inserted interactively, the name of the turrets (in which the tools used to machine those specific operations are stationed) will be suffixed to the Operation node name within parentheses. However, the turret name displayed within the parentheses will depend on the setting assigned to the *Turret Name* option in the context menu for the *Machine* node in the *Tools tree*.

  - If the *Turret Name* option is set to *Static Name*, then the default names of the turrets will be displayed within parentheses.
  - If the *Turret Name* option is set to *Label*, then the default names of the turrets will be displayed within parentheses.



Turret Dropdown list under Tool Tab of New Operation UI when turrent name is set to Label

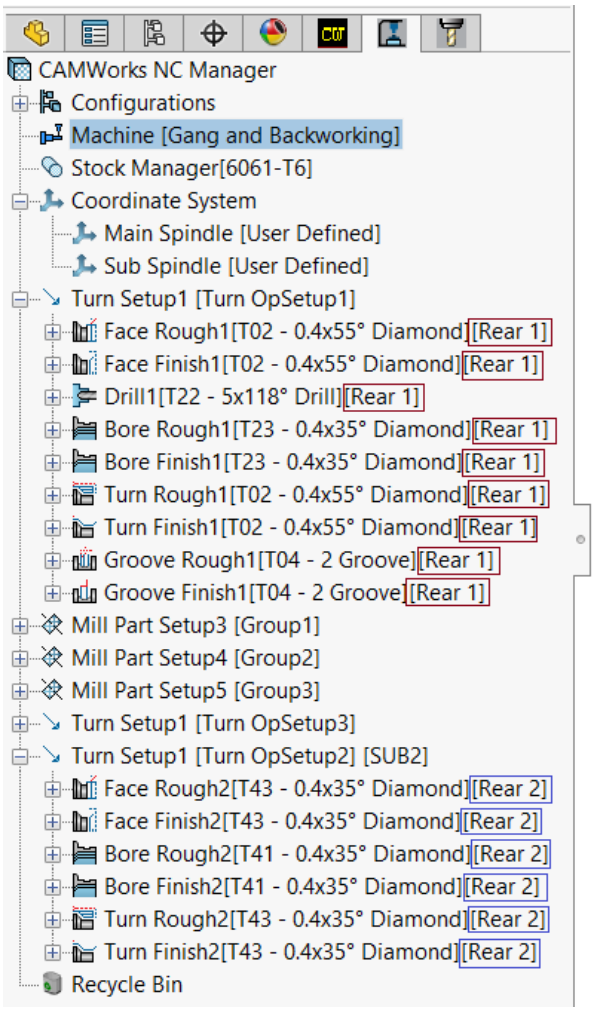


Turret Dropdown list under Tool Tab of New Operation UI when turret name is set to Static Name

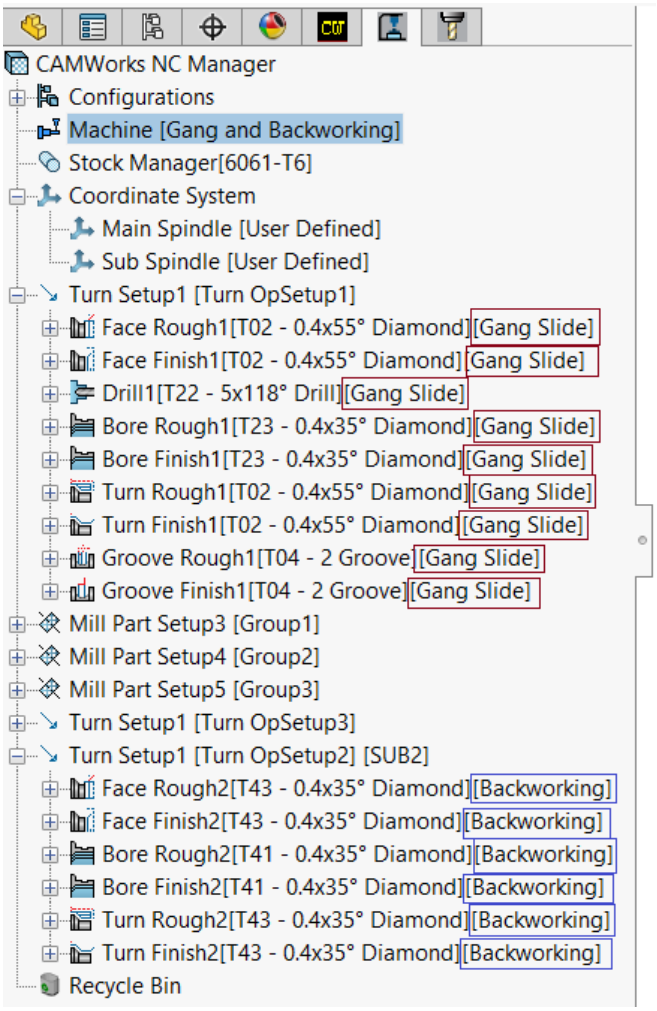
- **Turret names assigned as suffixes to Operation nodes in the Operation Tree**

  Whenever new operations are generated using Generate Operation Plan command or inserted interactively, the name of the turrets (in which the tools used to machine those specific operations are stationed) will be suffixed to the Operation node name within parentheses. However, the turret name displayed within the parentheses will depend on the setting assigned to the *Turret Name* option in the context menu for the *Machine* node in the *Tools tree*.

  - If the *Turret Name* option is set to *Static Name*, then the default names of the turrets will be displayed within parentheses.
  - If the *Turret Name* option is set to *Label*, then the default names of the turrets will be displayed within parentheses.



Turret Names Suffixed to names of Operation Nodes when 'Turret Name' Option is set to 'Static Name'



Turret Names Suffixed to names of Operation Nodes when 'Turret Name' Option is set to 'Label'

Tools

Cutting Parameters for Turn Inserts Based on Stock Material

Purpose:

To provide the ability to define cutting parameters for Turn Inserts based on the material of the stock being machined

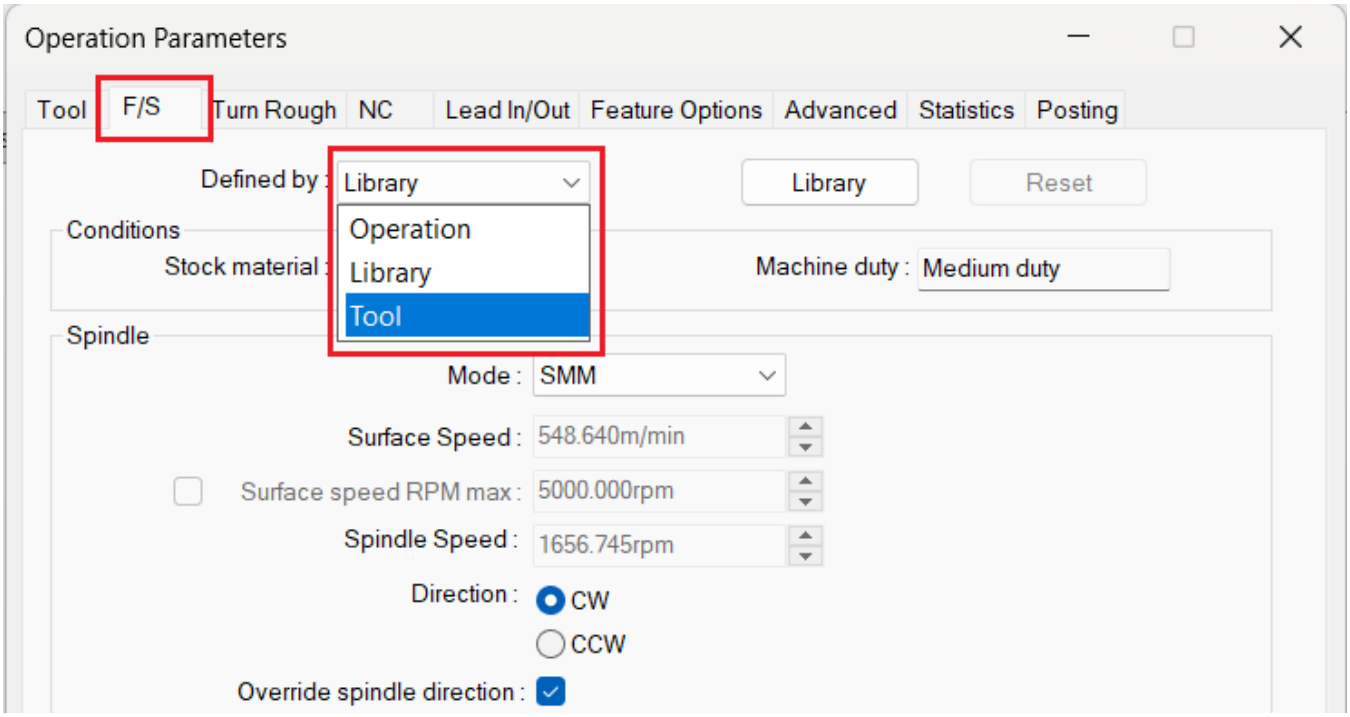
Implementation:

The Functionality of Defining Cutting Parameters for Tools

In the previous version of CAMWorks, the functionality to define cutting parameters for Mill tools based on the stock material being machined is available. From *CAMWorks 2026* version onwards, this functionality has been extended to Turn Inserts.

For each Turn Insert, one or more Stock Material Groups can now be defined and cutting parameters can now be defined for Turn Inserts based on the stock material to be machined. Every Stock Material Group defined for a Turn Insert will have a set of cutting parameters associated with it that can be defined/edited in the *Cutting Parameters* dialog box (both in the Technology Database as well as the CAMWorks application).

Under ***F/S*** tab of the *Operation Parameters* dialog box, a new option named ***Tool*** is now available in the ***Defined by*** dropdown list. When this ***Tool*** option is selected in the *Defined by* dropdown list, *CAMWorks* will use the feed and speed values defined in the *Cutting Parameters* dialog box. (The default setting for the ***Defined by*** dropdown list can be assigned in the Technology Database.)



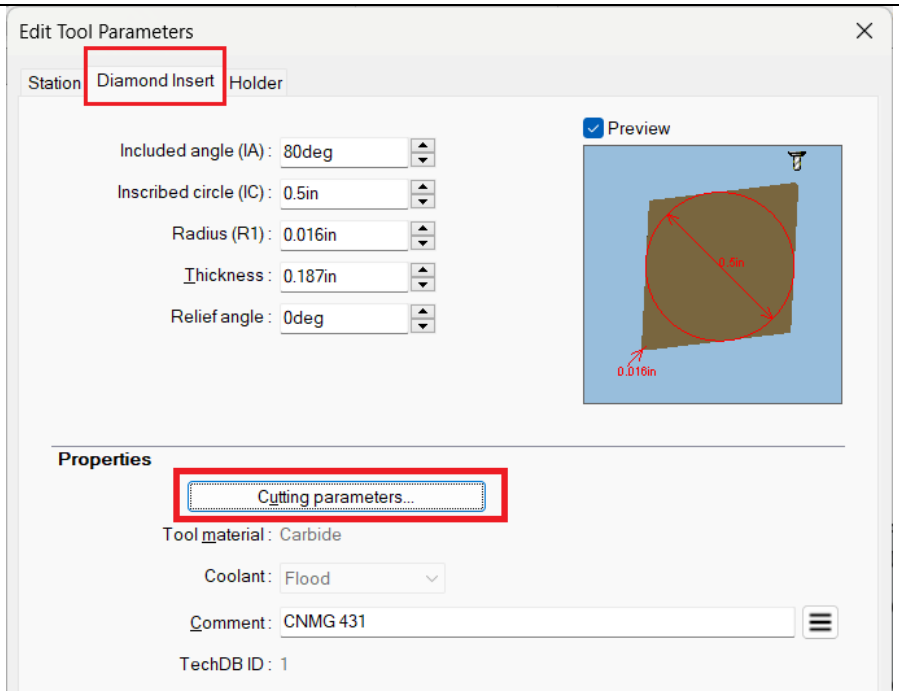
New 'Tool' Option introduced in the 'Defined By' Dropdown list under F/S tab

New 'Cutting Parameters' Dialog Box in CAMWorks for Turn Inserts

Cutting Parameters can now be defined for all types of Turn Inserts. However, these cutting parameters will be applied only when ***Tool*** option is selected in the *Defined by* dropdown list under *F/S* tab of the *Operation Parameters* dialog box for a Turn operation.

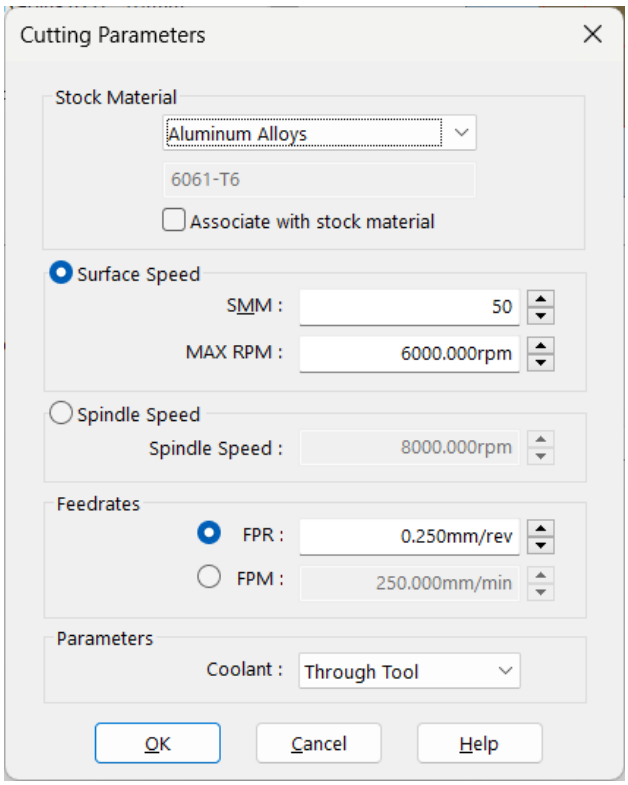
A new ***Cutting Parameters*** button is in the following user interfaces:

- In the *Turn Insert* page under the ***Tool*** tab In *Operation Parameters* dialog box
- Under the Turn Insert page in Edit Tool Parameters dialog box



Cutting Parameters button introduced in Turn Insert tab

Clicking on the ***Cutting Parameters*** button will display the ***Cutting Parameters*** dialog box.



Cutting Parameters Dialog Box

- **Cutting Parameters when the Associate with Stock Material' checkbox is checked**

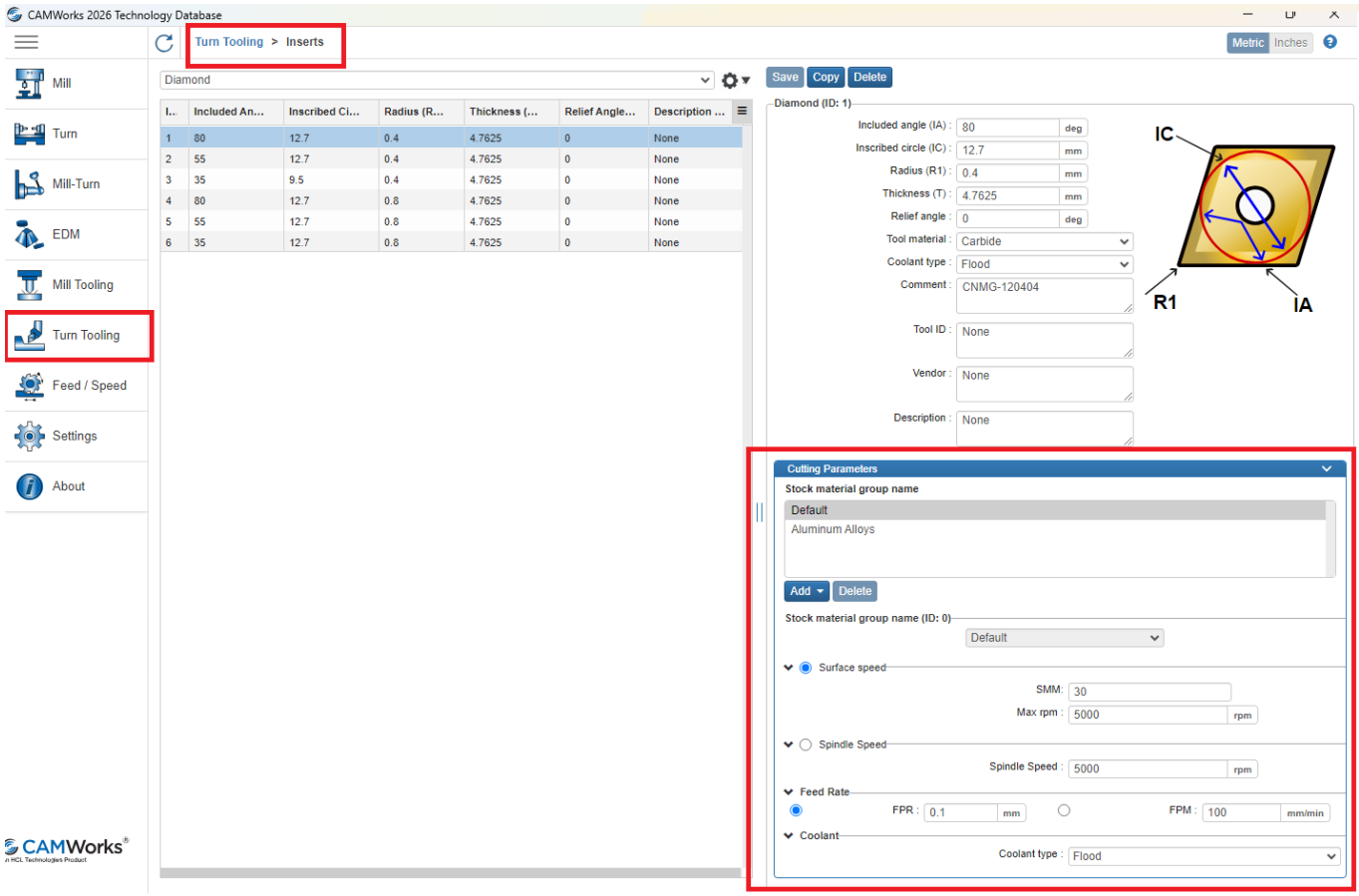
  By default, the ***Associate with Stock Material*** checkbox option within this dialog box will be checked. When checked, the ***Stock Material group*** will be automatically selected such that it is in sync with the Stock Material defined in the *Stock Manager* dialog box. The cutting parameters associated with the selected *Stock Material Group* will be populated within their respective fields within the *Cutting Parameters* dialog box. These parameters associated with *Surface speed*, *Spindle speed* and *Feedrates* will be disabled once populated.

- **Cutting Parameters when the Associate with Stock Material' checkbox is unchecked**

  When unchecked, all the parameters associated with *Surface speed*, *Spindle speed*, *Feedrates*, and *Coolant* within this dialog box will be enabled. Select the desired *Stock Material Group* and then use these parameters to define/edit the cutting parameters.

Defining Default 'Cutting Parameters' for Turn Inserts in Technology Database

In the Technology Database, a new *Cutting Parameters* group box has been introduced in the ***Inserts*** form under ***Turn Tooling*** menu for all types of Turn Inserts. Use this form to define the default cutting parameter values that will populate the corresponding *Cutting Parameters* dialog box within the *CAMWorks* application when the Turn Insert is selected to machine a stock material.



'Cutting Parameters' Group Box introduced in Turn Inserts Form within Technology Database

.

Option to Assign Tool Offset, Compensation IDs, and Tool Shift Parameters

Purpose:

To provide the options to assign Tool Offsets, Compensation IDs, and Tool Shift Distance (distance between the face of the Guide Bush and the tools mounted on the station) to each station in the Tool Crib/Tool Post

Implementation:

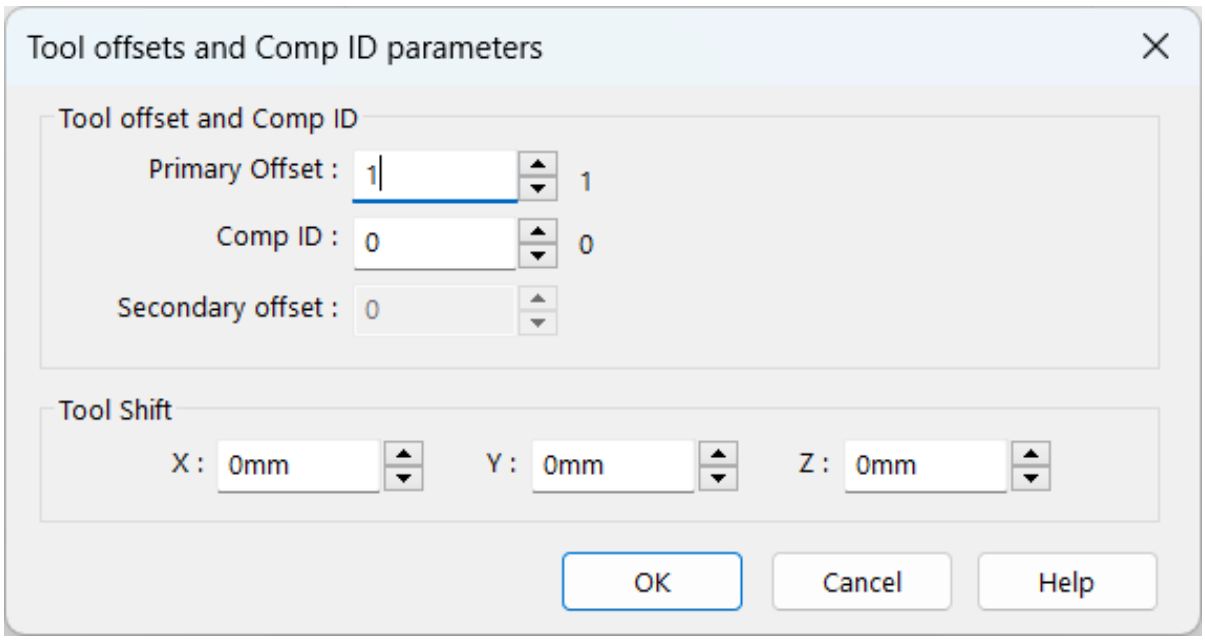
For Swiss-Turn machines, a tool on an assigned station will have offsets and Compensation IDs. Depending on the operation these values can be overridden. Additionally, some Tool Posts on such machines do not have any movement along the Z axis. The tools mounted on such Tool Posts are set at a defined distance from the face of the Guide Bushing. An option to define the Tool Shift Distance (distance between the face of the Guide Bush and the tools mounted on the station) needs to be available.

From *CAMWorks 2026* onwards, CAMWorks allows assigning Tool Offsets, Compensation IDs, and Tool Shift Distance for each station. These can be used in the posted output to meet various machining requirements.

A new dialog box named "Tool Offset and Comp ID Parameters" has been introduced to address these advanced Tool Crib/ Tool Posts settings. The *Tools Offset and Comp ID Parameters* dialog box appears when you click the *Tool Offset and Comp ID Parameters* ... command button below the Gage Offset parameters under the *Station Page* of the *Tools tab* in the *Operation Parameters* dialog box.

The *Tools Offset and Comp ID Parameters* dialog box can be used to assign settings for the following parameters:

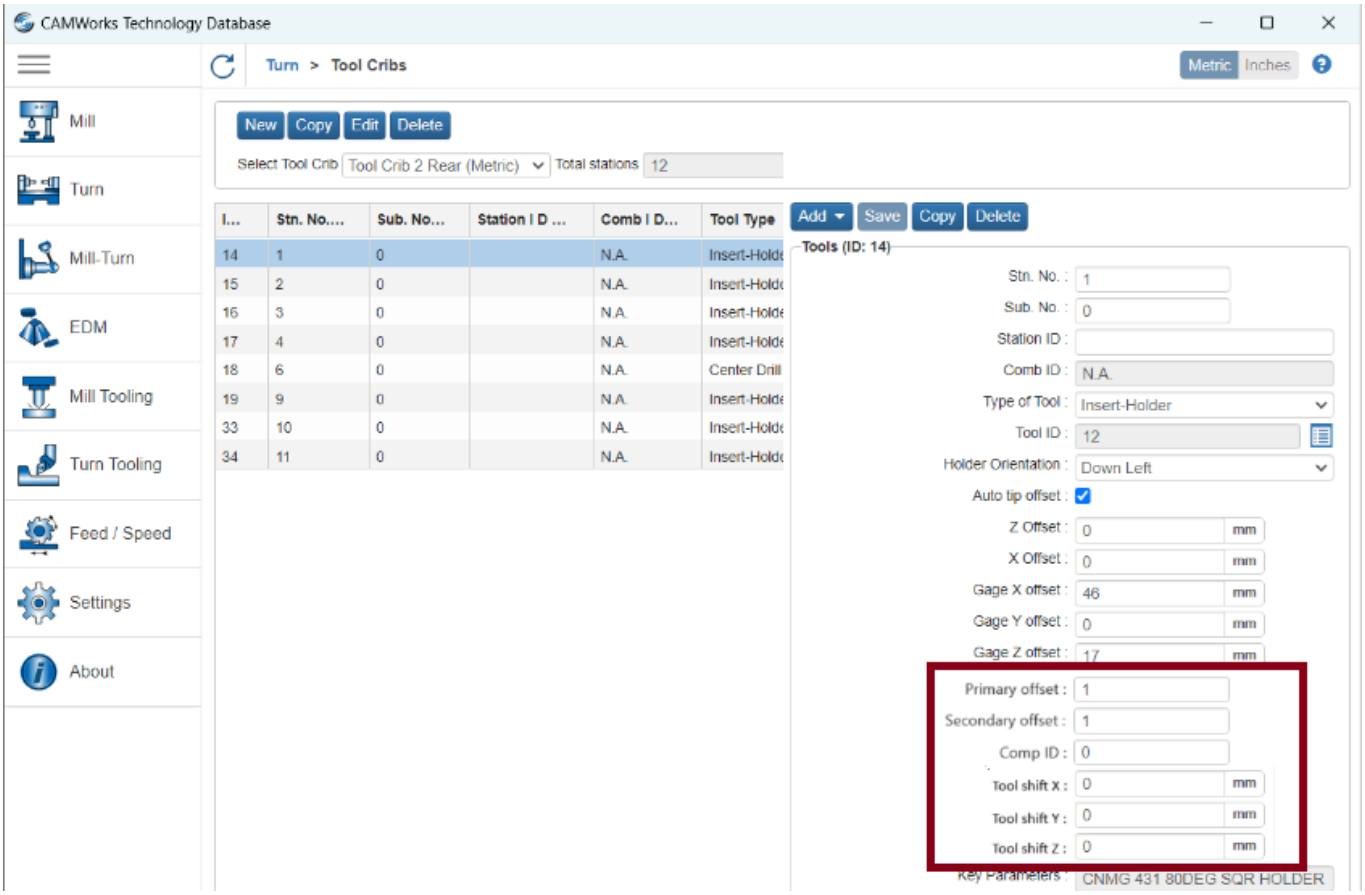
- **Primary Tool Offset**
  Use this parameter to assign the primary offset for the selected station. Also, different *Primary Offset* values for different operations using the same tool assigned to a specific station applicable to the selected operation.
- **Compensation ID**
  Use this parameter to assign the compensation ID for the selected station, applicable to the selected operation.
- **Secondary Tool Offset**
  Use this parameter to assign the secondary offset for Groove Insert Tools.
- **Tool Shift Distance**
  Use these parameters to define the distance between the face of the Guide Bush and the tool at the selected station.



The new Tool Offsets and Comp ID Parameters dialog box

Defining the default values for the Primary Offset, Secondary Offset, Comp ID, and Tool Shift distance in TechDB

The default values for these parameters can be saved in the corresponding fields of the Turn Tool Crib and Mill-Turn Tool Crib forms within TechDB.



Newly added Tool Offset, Comp ID, and Tool Shift parameters in the Tools form under Turn Tool Cribs.

Spindle Association with Tool Station to Define Work Area

Purpose:

To allow users to define a particular spindle as the Work Area for each tool present in the active Tool Crib so that the tool can machine only on the assigned spindle(s)

Implementation:

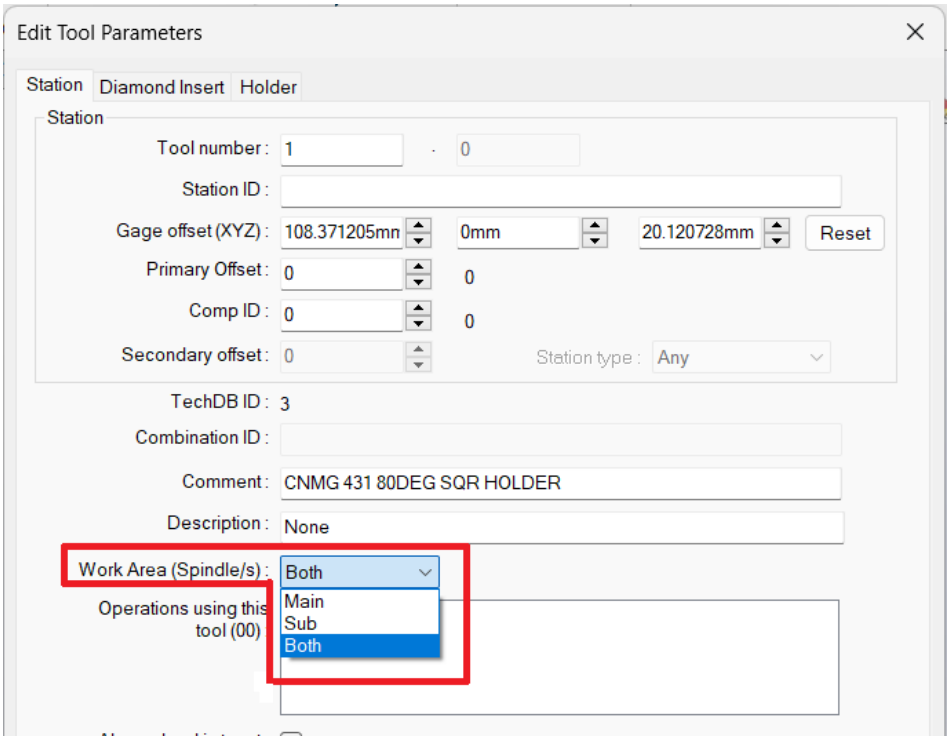
A new parameter of ***Work Area (Spindle/s)*** has been added on ***Station*** tab for all tools for Turn, Mill-Turn and Multi-Function Lathes. Use this dropdown list to define the spindle(s) that will be allowed to use the selected Tool Station.

The spindle options available in this dropdown list are:

- Main
- Sub
- Both (Default option)

Note:

For all legacy parts programmed using a previous of CAMWorks, the ***Work Area (Spindle/s)*** property will be set to ***Both*** for all operations.



Work Area (Spindle/s) Dropdown List introduced in Station Page under Tool Tab

How Work Area (Spindle/s) Property Impacts The Tool Selection For Operations

- Every station in the active Tool Crib will be assigned a Work Area (Spindle)
- When Generate Operation Plan is done, then the tool on a station with a satisfying work area condition is selected for each operation. (All the other rules to filter the correct tools viz. tool diameter, material are executed before selecting the tool from the active tool crib)
- In case if a new tool has to be added due to the result of Generate Operation Plan command then, it will be added to the appropriate station having a suitable work area to machine the selected feature.
- While interactively inserting the operations, when a feature is selected then, based on the spindle it is being machined, the available tool list will be filtered to display only tools which have the appropriate work area property.

Option to Create Empty Tool Stations in Tool Cribs and associated Functionalities

Purpose:

- To provide the option to define empty tool stations in a tool crib in CAMWorks
- To provide an option to assign Station Type for Tool Stations in Mill-Turn Tool Cribs
- To provide an option to remove a tool assigned to a Tool Station, leaving the tool station empty
- To provide the option to create a new tool crib in TechDB with a user-specified number of empty tool stations that can later be populated
- To provide an option that prevents new tool stations from being added to the selected tool crib
- Addition of new tools to active tool crib when the *Generate Operation Plan* command is executed

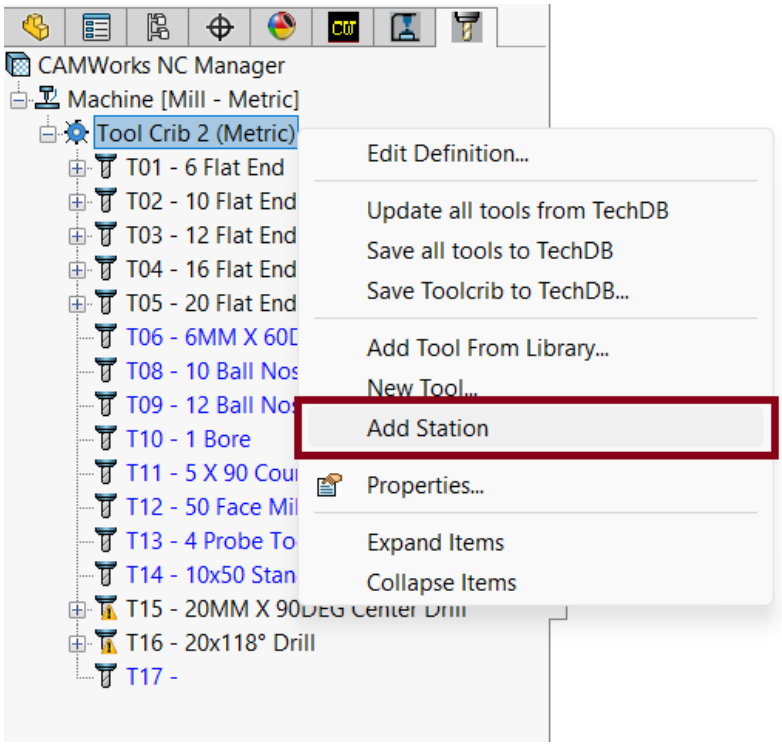
Implementation:

This functionality to create empty tool stations in tool cribs is especially beneficial for Swiss Turn machines that may have empty stations in tool posts or turrets.

From *CAMWorks 2026* version onwards, you can define empty tool stations under the CAMWorks Tools Tree and also in Tool cribs within the TechDB. A context menu option to remove tools from tool stations of an active Tool crib under the CAMWorks Tools Tree has also been introduced. This feature is particularly useful for machine configurations where not all stations need to be occupied, ensuring flexibility and efficiency in tool management.

Option to Define Empty Tool Stations in CAMWorks

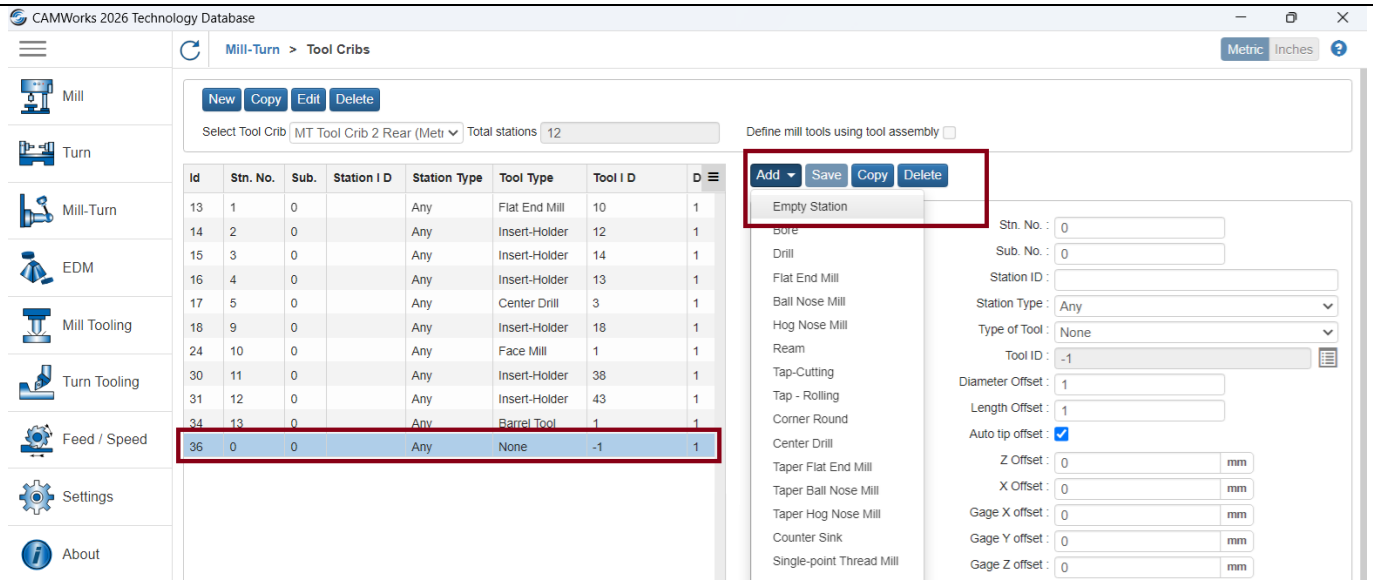
Under the CAMWorks Tools tree, a new context menu option named ***Add Station*** is now available. Executing this command will allow the addition of empty tool stations to the active Tool crib. The newly created empty station will be displayed as the last entry in the active Tool crib under the CAMWorks Tools Tree.



'Add Station' Option on Context Menu of Tool Node Under CAMWorks Tools Tree Tab

Defining Empty Tool Stations in TechDB

In the ***Tool Cribs*** form within the TechDB, an empty tool station can be added to the selected tool crib. Select the ***Empty Station*** option from the dropdown list displayed on clicking the ***Add*** button. The newly added empty tool station will be listed as the last entry in the selected Tool crib.

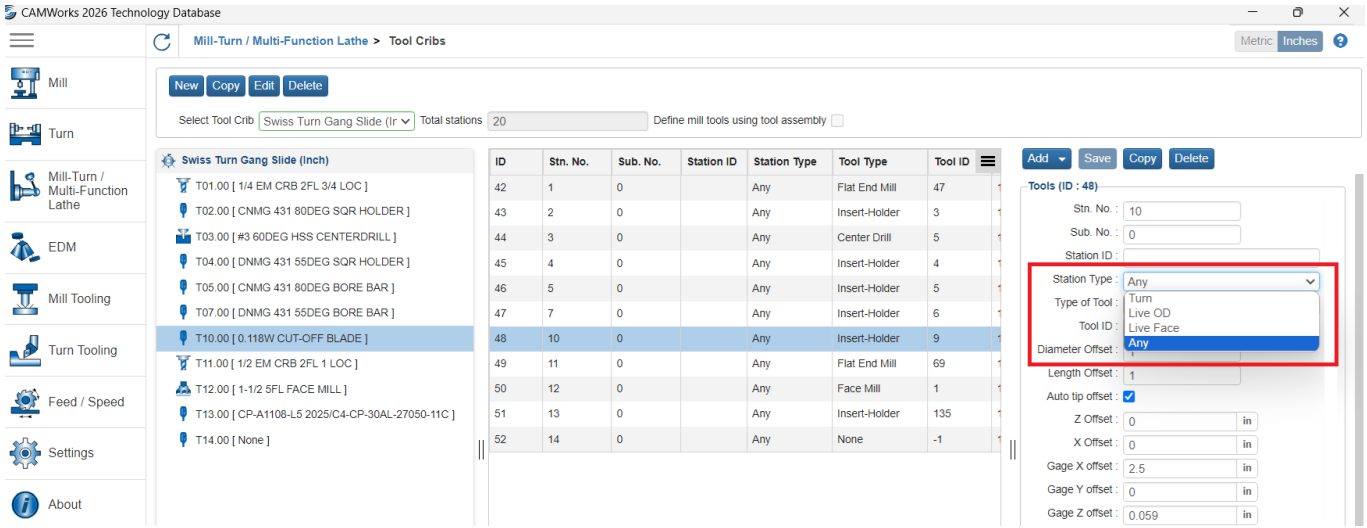


'Empty Station' Option for Add Button in Tool Cribs Form within TechDB

Assigning Station Type for Tool Stations in Mill-Turn Tool Cribs

Every tool station within a Mill-Turn/Multi-Function Lathe tool crib will now have a new property labelled ***Station Type***.

Within the TechDB, this property will be visible as the *Station Type* dropdown list is available in the ***Tools*** form of the *Mill-Turn/Multi-Function Lathe>Tool Cribs* form.

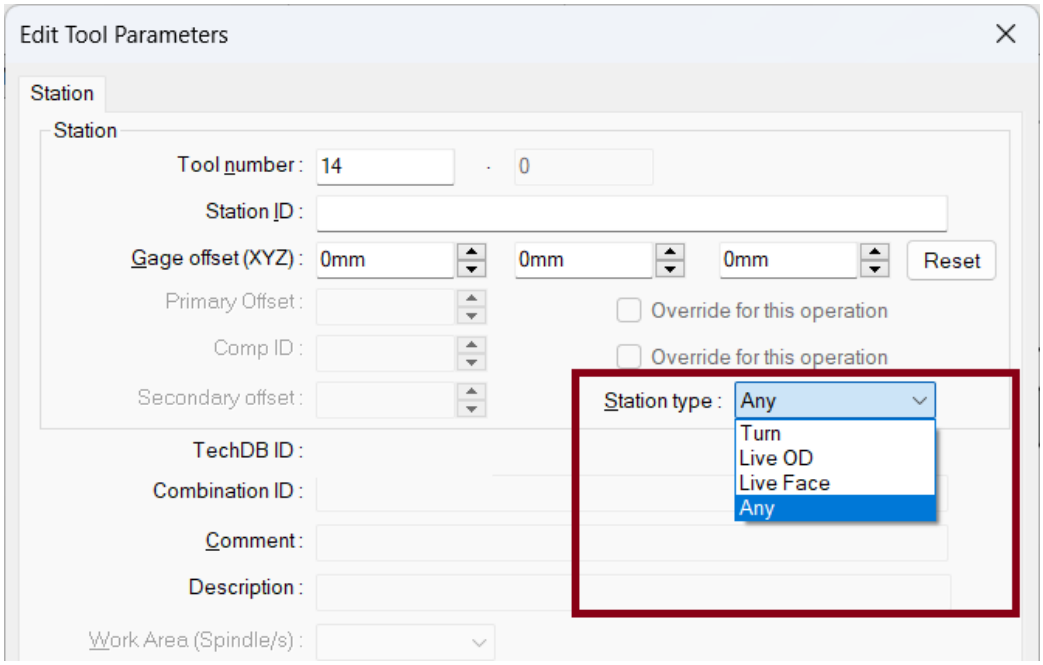


Station Type dropdown List in Tools Form of Mill-Turn/Multi-Function Lathe>Tool Cribs Form

Use this *Station Type* parameter to assign the station type which will indicate the type of tool that can be assigned to the specific station. The table given below indicates the tool types that can be assigned to a tool station based on the station type assigned to it using the *Station Type* dropdown list. These tool types will be displayed in the *Type of Tool* dropdown list in the *Tools* form of TechDB.

		Options available in the 'Type of Tool' dropdown list
1.	Station Type 'Any'	None, Insert-Holders, Mill Tools, Single Point Tools, User Defined Tools
2.	Station Type 'Turn'	None, Insert-Holders, Drill, Center Drill
3.	Live OD	None, Mill Tools, Single Point Tools, User Defined Tools
4.	Live Face	None, Mill Tools, Single Point Tools, User Defined Tools

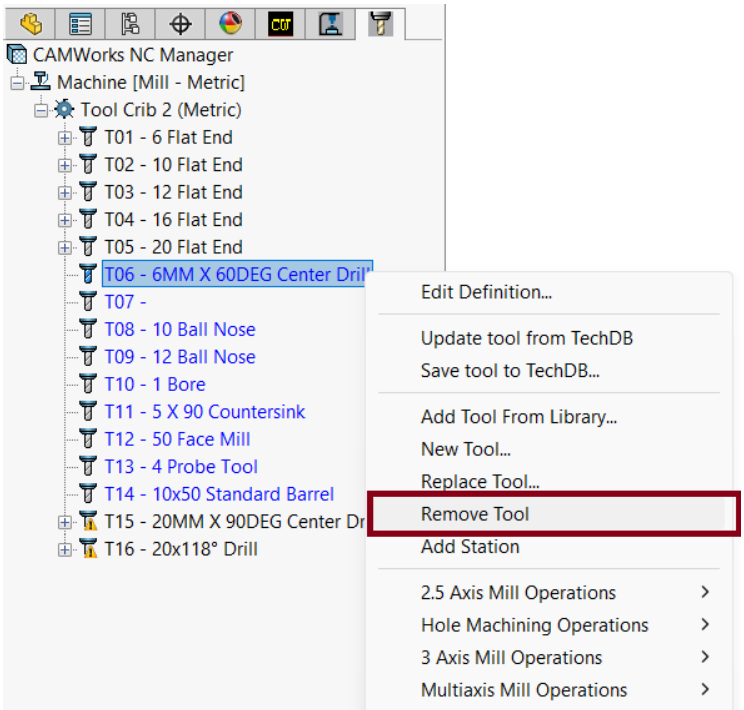
Within the CAMWorks application, an empty Tool Station can be added to an active tool crib by selecting the ***Add Station*** command in the context menu of a Tool node or Tool Crib node under the Tools tree. The ***Edit Tool Parameters*** dialog box is displayed on executing this command. Under the ***Station*** page of this dialog box, the Station type dropdown list will be displayed if the active machine configuration is a Mill-Turn/Multi-Function Lathe. Within this dialog box, use this dropdown list to indicate the station type for the new empty station being added to the active tool crib.



'Station Type' Dropdown list under Station Tab of 'Edit Tool Parameters' Dialog Box

Removing Tools from Tool Station Nodes

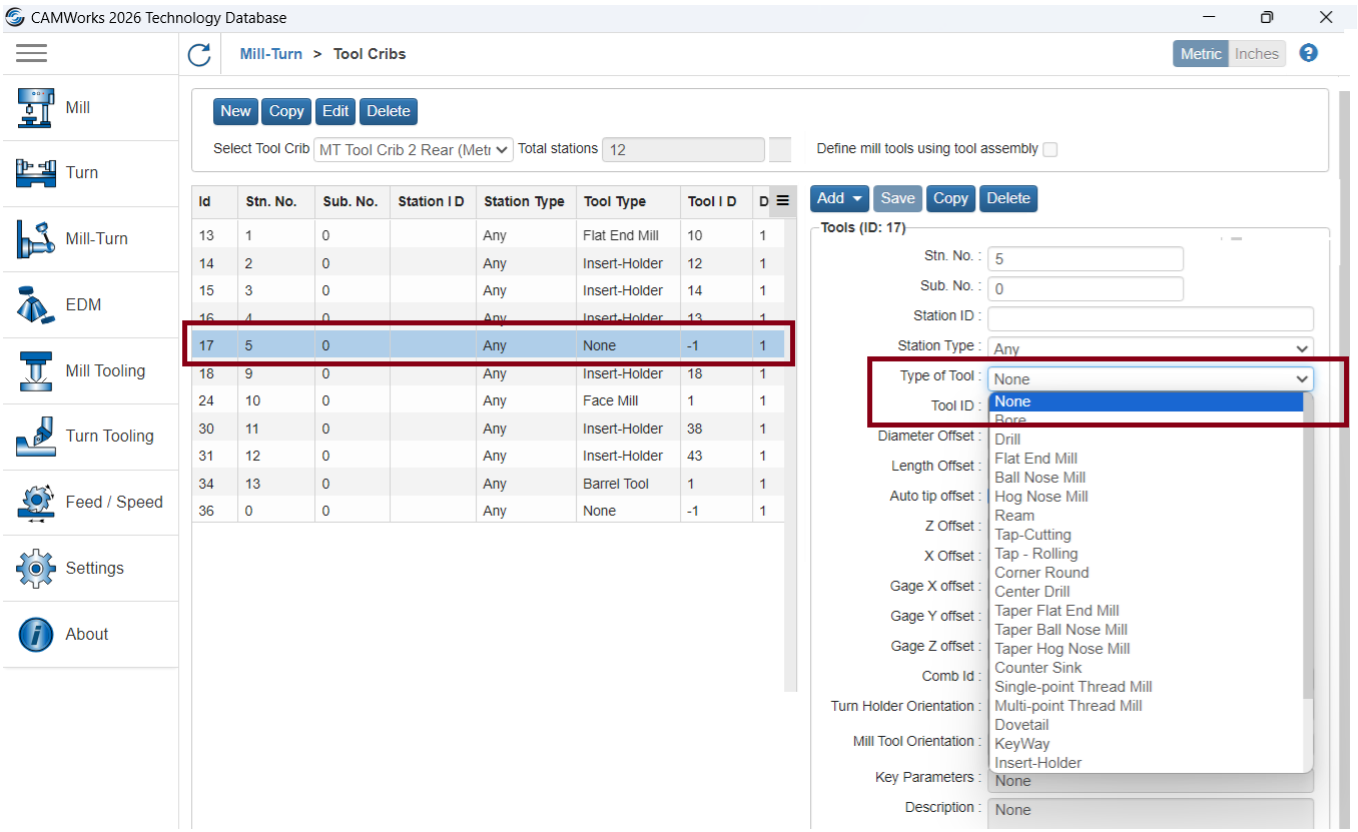
A new context menu option ***Remove Tool*** is now available for Tool nodes under the ***CAMWorks Tools tree***. It will be available in the context menu of only those tool nodes that are not in use by any of the operations. Executing this command will remove the tool from the selected tool station node, leaving it empty. The station node will be displayed as empty both under the *CAMWorks Tools Tree* as well as the *Tool Crib* tab under *Operation Parameters* dialog box.



'Remove Tool' Option on Context Menu of Tool Node Under CAMWorks Tools Tree Tab

Removing Tools Assigned to Tool Station Nodes in TechDB

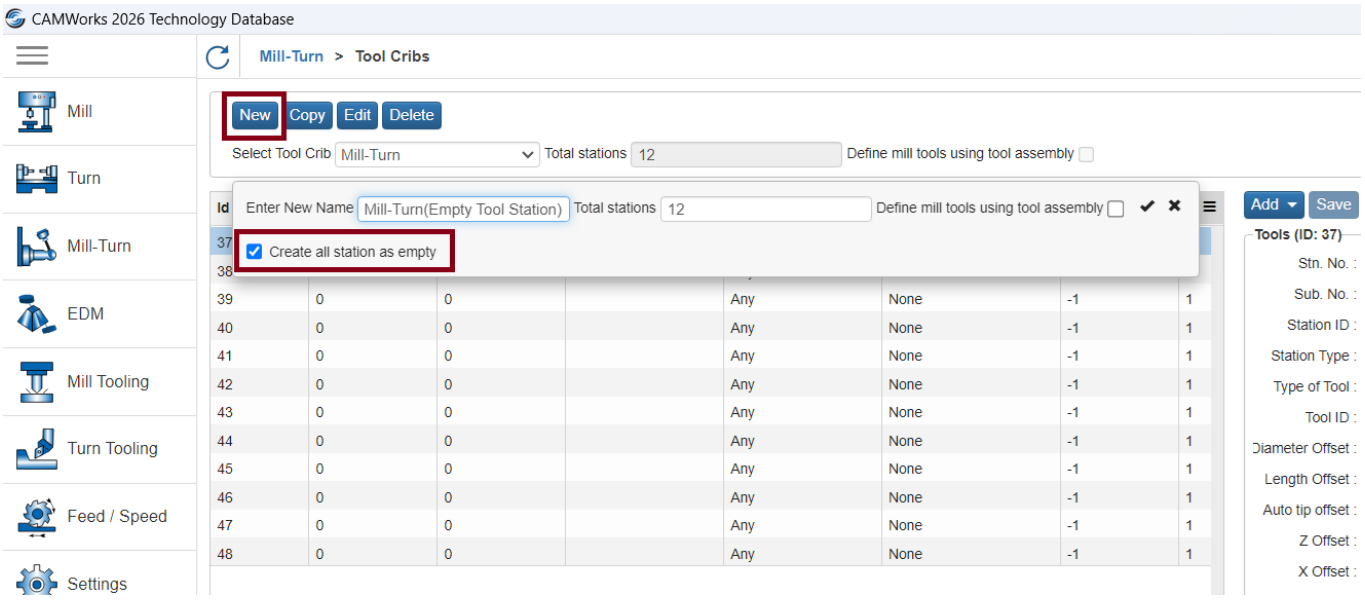
Within the *Tool Cribs* form of the TechDB, to remove a tool from a tool station of the active Tool crib, select '***None***' in the ***Type of Tool*** dropdown list within the ***Tools*** Form. This will set the Tool ID parameter to **-1**, indicating successful tool removal.



'None' Option in Type of Tool dropdown list within Tools Form of Tool Cribs Form in TechDB

Creating a New Tool crib in TechDB with User-specified Number of Empty Tool Stations

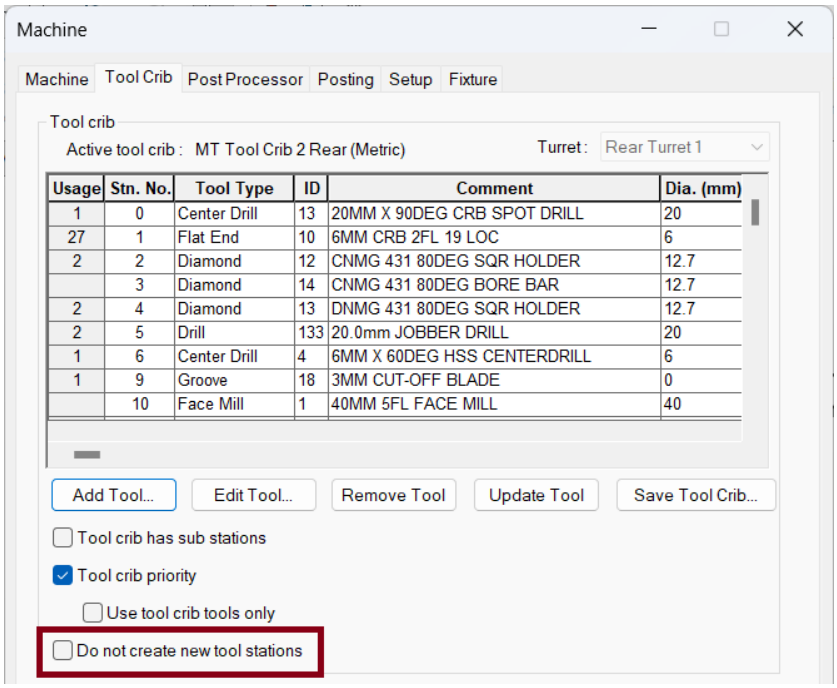
In *Tool Cribs* form of TechDB, a new checkbox option ***Create all stations as empty*** is available in pop-up for creating new Tool Cribs. When this checkbox option is checked, all the tool stations in the newly created tool crib will be empty. The number of empty tool stations created will be based on the number assigned in the Tool Stations field within the pop-up.



'Create all stations as empty' Checkbox Option in Tool Cribs Form of TechDB

Option to prevent New Tool Stations from being added to selected Tool Crib

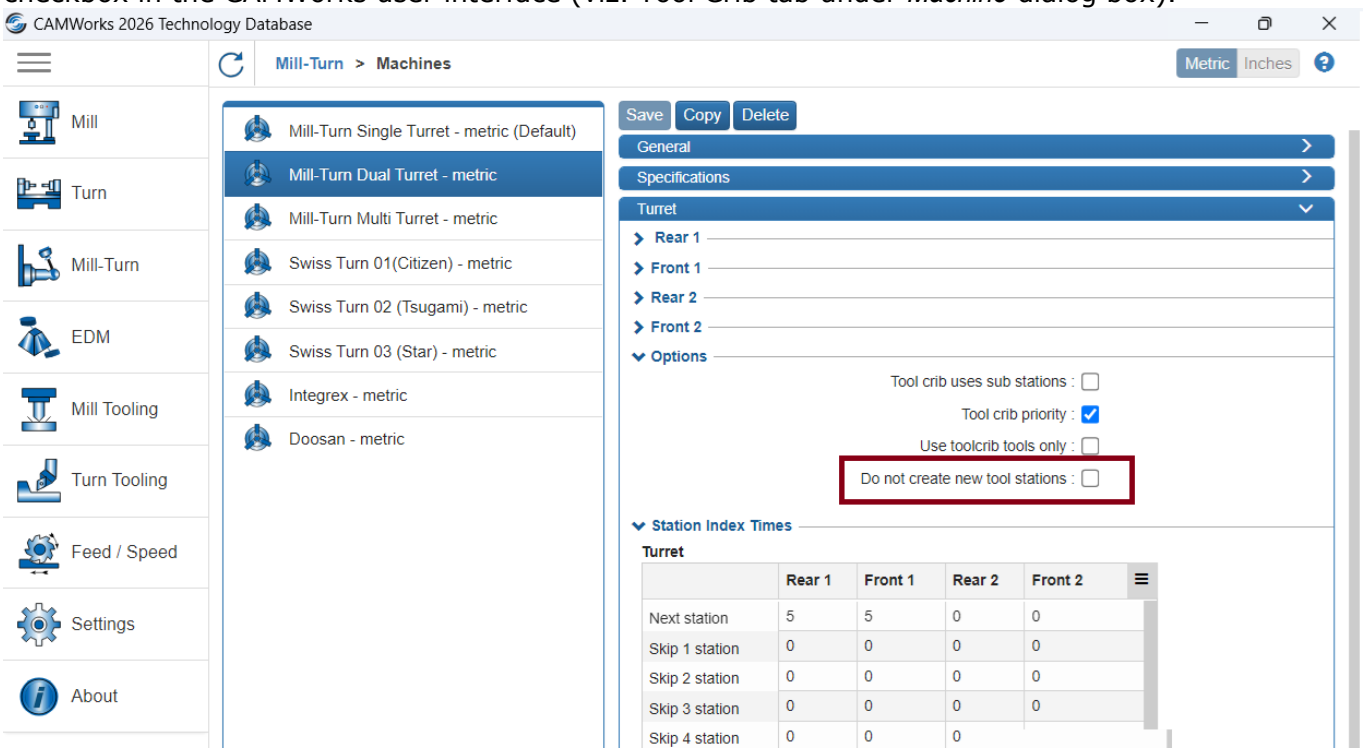
A new checkbox option ***Do not create new tool stations*** is now available under the *Tool Crib* tab of the *Machine* dialog box. When checked, CAMWorks will not add a new tool station to the active Tool crib. This option will be disabled when both the *Tool Crib priority* and *Use toolcrib tools only* checkbox options are checked. The default for this checkbox can be defined in TechDB.



'Do not create new tool stations' checkbox Option under Tool Crib Tab of Machine Dialog Box

Option to prevent New Tool Stations from being added to selected Tool Crib in TechDB

Within the TechDB, the ***Do not create new tool stations*** checkbox option is available under the ***Turret*** tab of the ***Machines*** form. Use this option to assign the default setting for the corresponding checkbox in the CAMWorks user interface (viz. Tool Crib tab under *Machine* dialog box).



'Do not create new tool stations' Checkbox Option under Turret Tab of Machines Form

Addition of New Tools to Active Tool Crib when the 'Generate Operation Plan' Command is Executed

When the *Generate Operation Plan* command is executed, new tools will be added to the tool crib from the tool library if they are required for machining the operations.

- For Mill and Turn tool cribs, these new tools will be assigned to the empty tool stations provided they are compatible with the Tool Type property of that empty tool station.
- For Mill-Turn tool cribs, these new tools will be assigned to the empty tool stations provided they are compatible with the *Station Type* and *Tool Type* property of that empty tool station.

Orientation for Mill Tools in Mill-Turn Tool Cribs

Purpose:

To provide an option to define the *Front* or *Back* orientation for Mill Tools in Mill-Turn tool cribs

Implementation:

In previous versions of CAMWorks, the option to define orientation for Mill Tools were *Left*, *Right*, *Up*, and *Down*.

From CAMWorks 2026 version onwards, Mill-Turn/Multi-Function Lathe machines having sliding heads are supported. When parts/assemblies are machined on such machines, the tools can approach the spindle from the Y axis direction as well. Previous version of CAMWorks had no option available to assign the orientation for such tools.

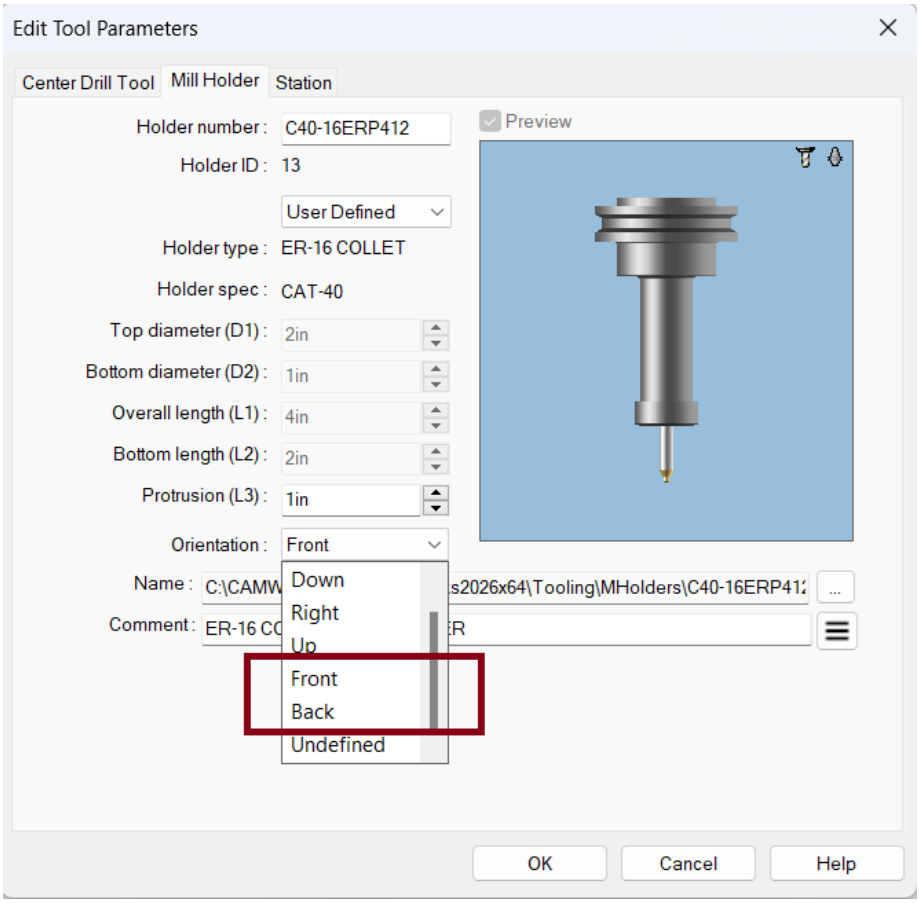
To address this limitation, for mill toolpaths machined on such machines having sliding heads, the Mill tool orientation can now be defined for tools approaching from the Y axis direction.

Under the ***Mill Holder*** tab of a selected Mill tool, the following two newly added options will be available in the Orientation dropdown list:

1. Front
2. Back

Note:

The ***Front*** and ***Back*** options will be available in the ***Orientation*** dropdown list only if the active machine is a Mill-Turn/Swiss Machine/Multi-Function Lathe machine with a sliding head.



Newly added 'Front' and 'Back' options in Orientation dropdown list under Mill holder tab for a Mill tool in a Mill-Turn Tool Crib

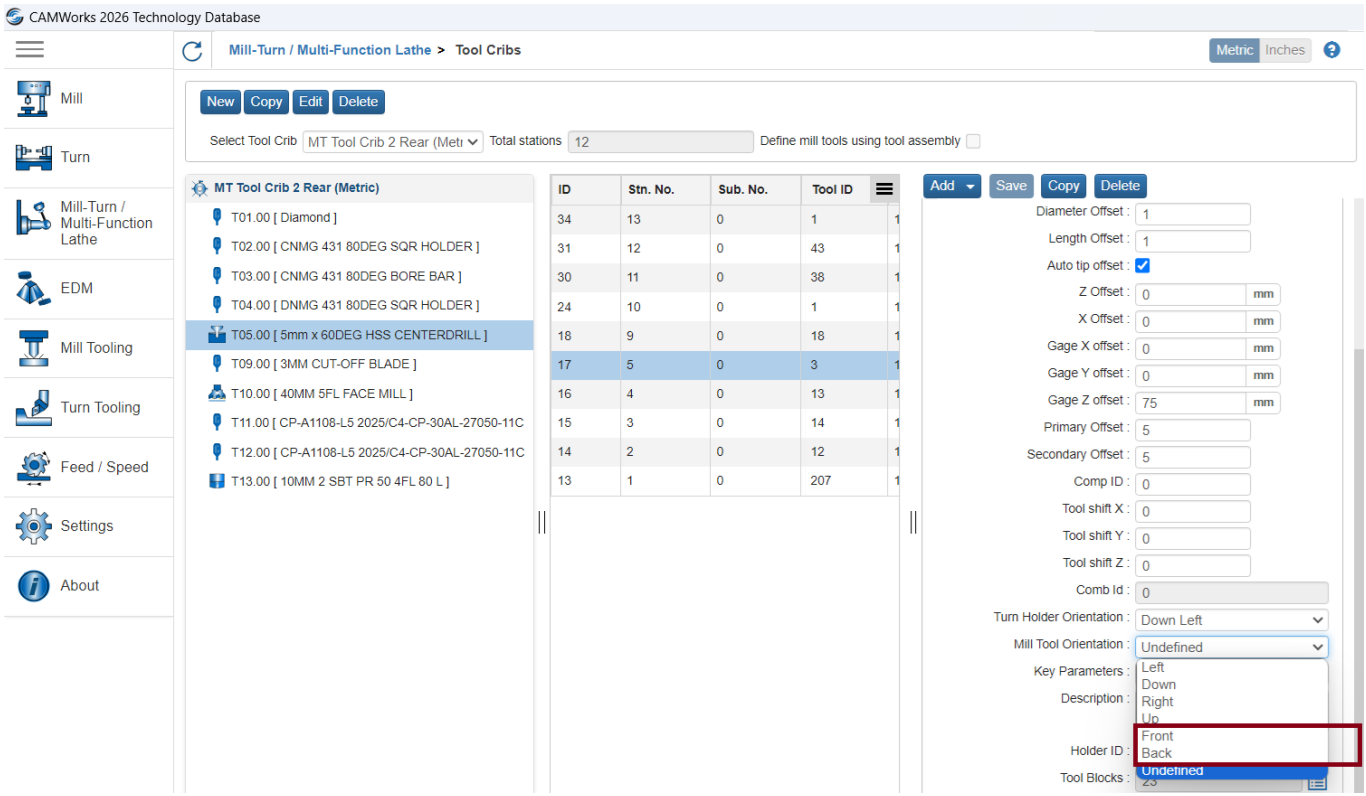
Factors affecting Orientation of Mill Tools assigned to Operations

The table given below indicates the proper orientation that will be assigned to a Mill tool in the Mill-Turn tool crib when the tool approaches from the Y axis direction:

Spindle	Turret	Orientation for Feature Location on OD
Main Spindle	Rear	Back
Main Spindle	Front	Front
Sub Spindle	Rear	Back
Sub Spindle	Front	Front

Assigning Default Orientation of Mill Tools present in Mill-Turn Tool Cribs saved in TechDB

Use the ***Mill-Turn/Multi-Function Lathe >Tool Cribs*** form of the TechDB to assign the default Mill Holder orientation for tools in tool cribs to be assigned to *Mill-Turn/Multi-Function Lathe* machines.



New 'Front' and 'Back' options under Mill Tool Orientation dropdown list in Tools Form of Mill-Turn/Multi-Function Lathe>Tools Cribs Form in TechDB

New Tool Changer Option 'Slide' available For All Machine Configurations within TechDB

Purpose:

To provide the option of new ***Changer*** method of ***Slide*** within *Turret tab* of Machine Configuration form within the Technology Database

Implementation:

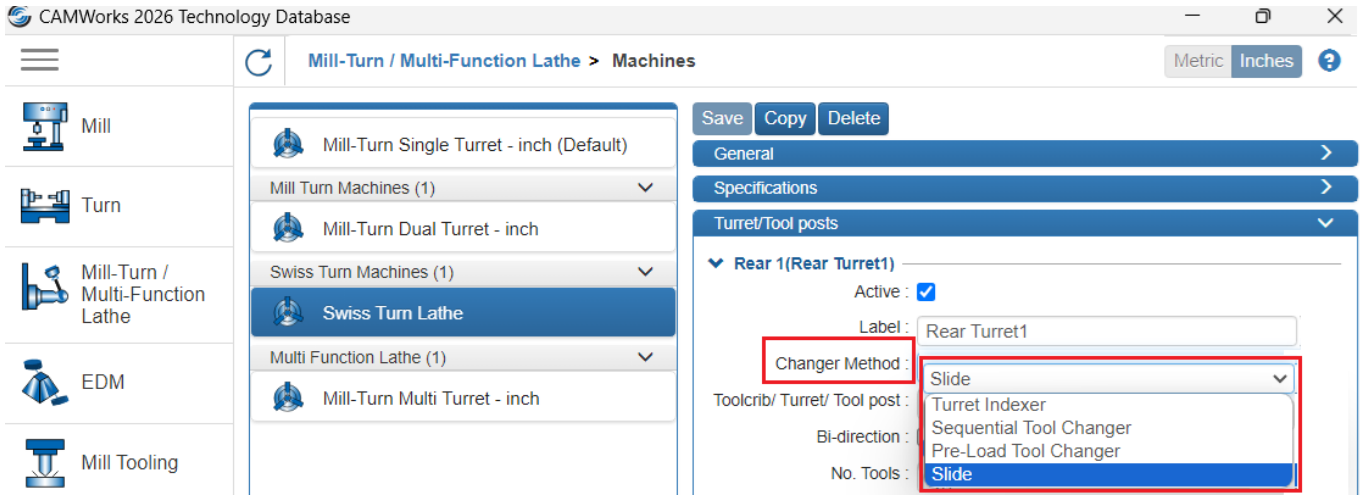
The ***Tool Changer*** for a CNC machine is a mechanical system that automatically switches cutting tools during the machining process. It enables these machines to perform multiple operations without manual intervention.

In the Technology Database, the controls for the tool changer are available as the ***Changer*** dropdown list in the following forms of the Technology Database:

- The ***Turret*** tab of the ***Mill Machines*** and ***Turn Machines*** forms within the TechDB
- The ***Turret/Tool Posts*** tab of the ***Mill-Turn/Multi-Function Lathe> Machines*** form within the TechDB

From *CAMWorks 2026* version onwards, an additional option named ***Slide*** has been made available in this dropdown list. Select this option when the tools in the selected tool post/turret slide to toggle to the cutting location.

This tool changer method is ideal for Swiss type lathes and compact CNC machines that require faster tool changes.



New 'Slide' option in 'Changer' Dropdown List under Turret/Tool Posts tab of the Mill-Turn/Multi-Function Lathe>Machines Form in TechDB

Operations and Toolpaths

Segmentation Manager for Turn Operations

Purpose:

To provide users with the option to split operations as per desired segment lengths and positions using a *Segmentation Manager* interface so that the part can be machined in multiple segments while being rigidly supported during the machining process

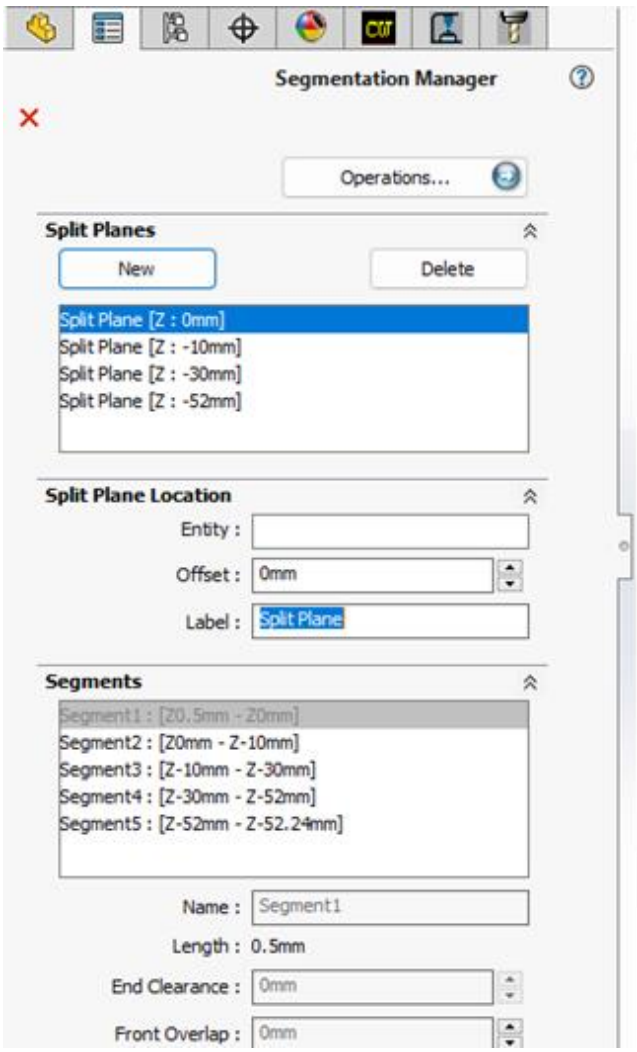
Implementation:

The Need for Segmenting to Machine Turn Features

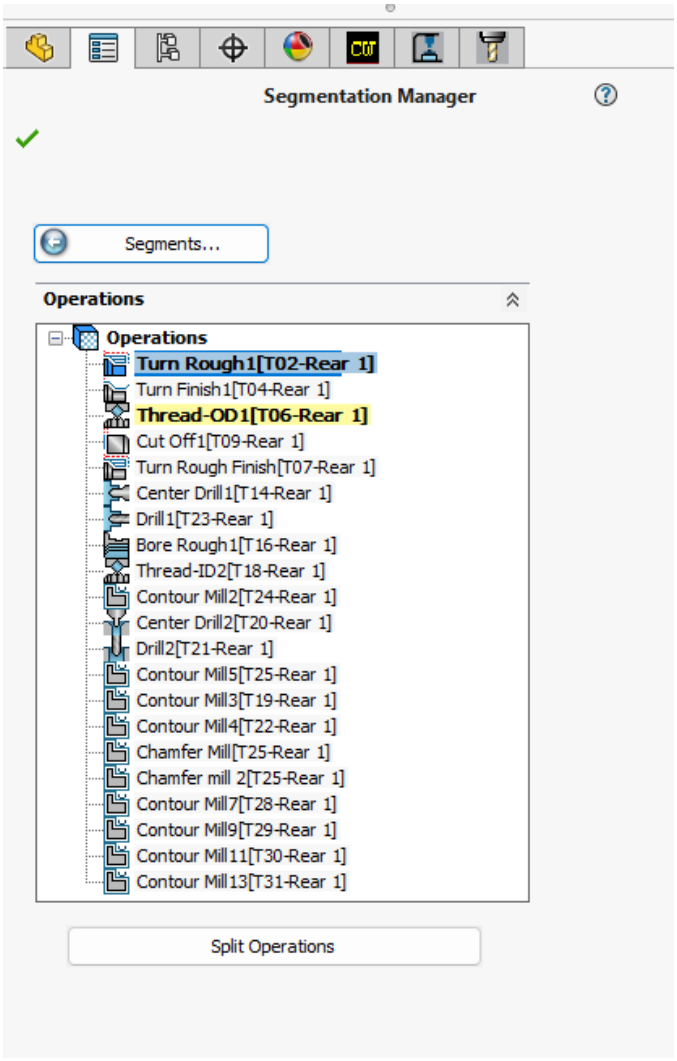
In Swiss Turning, the machining length of any Turn feature has to be less than equal to the carbide length of the guide bushing. If the feature being machined is longer than the Carbide Length, then the Turn feature needs to be machined in separate segments.

The Segmentation Manager Wizard

The newly introduced ***Segmentation Manager Wizard*** allows users to split Turn operations as per the segment lengths and positions. It is an interactive interface that allows users to perform segmentation for every Turn operation on the Outer Diameter as per their requirements. This wizard contains the *Segments* page and *Operations* page for defining segments and selecting operations for splitting, respectively.



Segments Page of Segmentation Manager Wizard

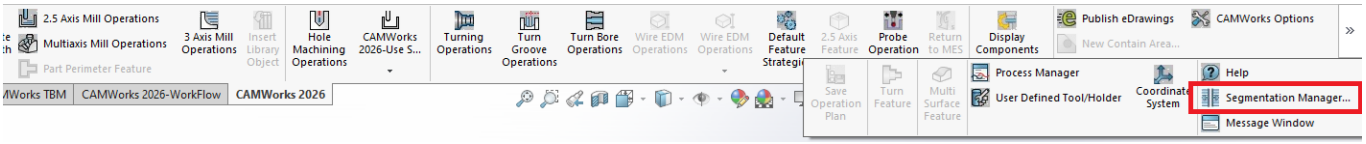


Operations Page of Segmentation Manager Wizard

Invoking the Segmentation Manager Wizard

The ***Segmentation Manager*** dialog box can be invoked by executing the following commands:

- By right-clicking on the *CAMWorks NC Manager* node in *CAMWorks Operation tree* and selecting *Segmentation Manager* option in its context menu.
- By clicking on the *Segmentation Manager* command of the *CAMWorks Command Manager*.



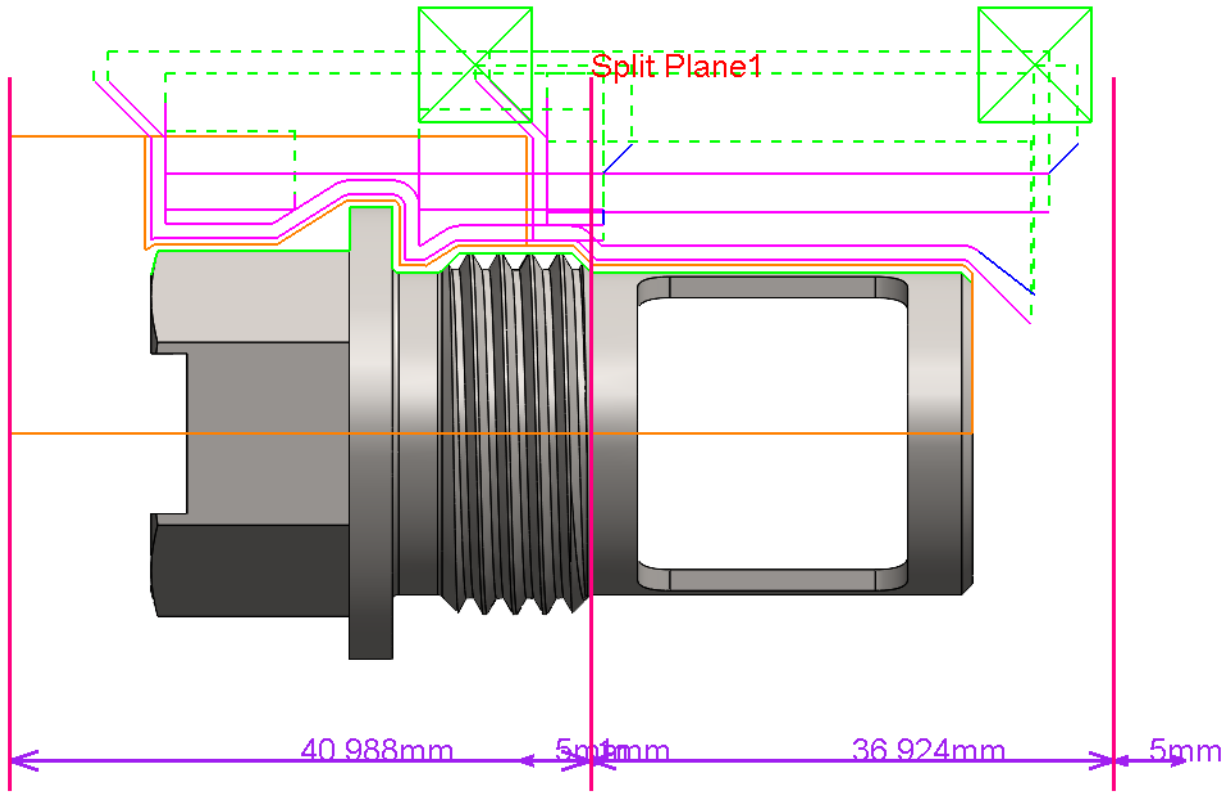
Segmentation Manager Command on the Command Manager

Segments Page of the Segmentation Manager Wizard to Split Turn Operations

In the ***Segments*** page of the *Segmentation Manager Wizard*, every time the ***New*** button is clicked, a split plane will be created and listed in the *Split Planes* list box. A corresponding segment will also be created and listed in the *Segments* list box. The 'segment' indicates the portion of the feature that will be machined by the operation. The split planes will be listed in the *Split Planes* list box in the descending order based on their proximity to the front of the part. They will also be displayed in the graphics area. Whenever you mouse hover over a split plane displayed in the graphics area, its corresponding entry within the *Split Planes* list box will be highlighted.

The location of a split plane can be defined by selecting a vertex/center point of an arc or circle along the profile of the part model. The name of the selected entity will be displayed in the ***Entity*** field.

For a split plane selected in the *Split Planes* list box, use the ***Offset*** field to define an offset value for the shifting it in Z direction. The offset distance will be measured from the assigned Entity.



Sample Turn toolpath limits split into segments by adding Split Planes

Operations Page of the Segmentation Manager Wizard

In the ***Segments*** page of the *Segmentation Manager Wizard*, when you click on the ***Operations*** button the ***Operations*** page of the *Segmentation Manager* Wizard will be displayed.

Operations Highlighted in Operations List Box within Operations Page

Within the *Operations* list box of the *Operations* page, all those operations that have their toolpaths spanning across one or more segment boundaries will be highlighted. The highlighted operations are the ones that are eligible for splitting into multiple operations based on the number of segments that they span across. When you mouse hover over these highlighted operations, a tooltip stating "Toolpath span is across ## segments" will be displayed. (The ## indicates the serial number of segments that the toolpath intersects with.)
To split one or more desired operations from the highlighted operations, select those operations and click on the ***Split Operations*** command button.

Split Operations Command in Operations Page of Segmentation Manager Wizard

The ***Split Operations*** command will be enabled when you select one or more highlighted operations from the *Operations* list box. Executing this command will split each selected operation into multiple operations based on the number of segments they span across. The number of split operations created for each operation directly corresponds to the number of segments it spans across.
For example, if a Turn operation spans across three segments and the *Split Operations* command is executed, then three split operations will be created.
Once this command is executed, each operation selected for splitting will be split as per the dimensions of the segments it spans across. For each such operation, all its newly split operations will be displayed as nested nodes under that operation in the *Operations* list box. When you highlight such a nested operation, its corresponding toolpath will be displayed in the graphics area.

Nomenclature for Split Operations

The nomenclature syntax for these nested operation nodes indicating split operations will be as follows:

<Operation Name><Segment #>[Tool Number - Turret Name]

Where:

- # indicate the segment number
- Turret name indicates the turret the tool is from (i.e., rear turret or front turret)

Update Operations Command in Operations Page of Segmentation Manager Wizard

The ***Update Operations*** command will be displayed in the *Operations* page when the dimensions of one or more segments are modified. In the *Segments* page of the *Segmentation Manager Wizard* after one or more operations listed in the *Operations* list box has been split. If a parent operation listed in the *Operations* list box has been split, then selecting that operation and executing the *Update Operations* command will rebuild the toolpaths of the split operations as per the updated segment dimensions.

OK Button of Segmentation Manager Wizard

After the desired operations are split/updated, click on the *OK* button to save the segmentation data and to implement these changes to the Operation tree.
When you click the *OK* button, *CAMWorks* will display the following message: "*Do you want to sequence the operations as per Z order?*"
There will be two options within this message box viz. *Yes* and *No*.

- If you select *Yes*, the *Segmentation Manager Wizard* will close, and all the newly split operations will be listed in the *Operation* tree as per the order of the segments. The toolpaths for this newly split operations will be generated/rebuilt.
- If you select *No*, the *Segmentation Manager Wizard* will close and all the newly generated operations will be listed in the *Operation* tree by replacing the original operation under the same setup. The toolpaths for this newly split operations will be generated/rebuilt.

Operations supported for splitting using Segmentation Manager Wizard

The following operations generated for OD Features are supported for segmentation within the *Segmentation Manager Wizard*:

- Turn Rough
- Turn Finish
- Groove Rough
- Groove Finish

Operations not supported for splitting using Segmentation Manager Wizard

The following operations are not supported for segmentation within the *Segmentation Manager Wizard*:

- Sub Spindle operations
- Probe operations
- Post operations
- Face operations
- Cut Off operations
- Suppressed operations
- Thread operations
- All operations generated for ID features
- VoluTurn operations (These are *Turn Rough* operations that have their *Method* set to *VoluTurn*)
- Prime Turing operations (These are *Turn Rough* and *Turn Finish* operations that have their *Method* set to *Prime Turning*)

Note:

If the ***Segmentation Manager Wizard*** is invoked after splitting the operations, then all the split operations will be listed in the *Operations* list box. Any split operations that are deleted but still present in the *Recycle Bin* will also be listed in the *Operations* list box. However, deleted operations present in the *Recycle Bin* that were not split will not be listed in the *Operations* list box.

New - Support for Bar Break Chamfering of the Stock in Turn Toolpaths

Purpose:

To provide the option of adding Bar Break moves to Turn Toolpaths generated for OD features in order to prevent/eliminate burrs that can potentially damage the Guide Bushings

Implementation:

During the Turn toolpath machining process, burrs (unwanted sharp edges) are formed by the tool inserts when machining the edges of the cylindrical stock. Burrs can cause potential damage when the stock material slides through Guide Bushings.

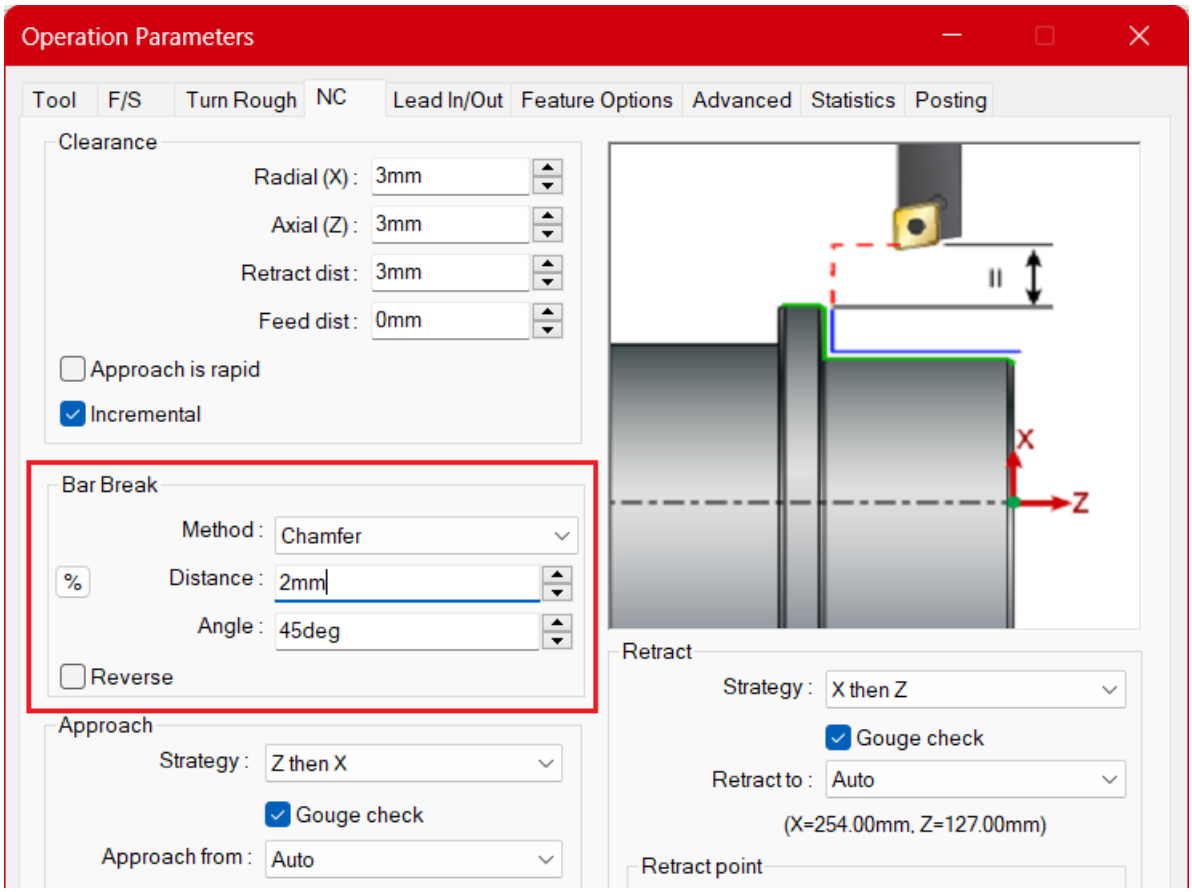
From *CAMWorks 2025* version onwards, to eliminate this issue of burr formation, the option to create additional moves called Bar Break moves has been provided for the following Turn toolpaths:

- Turn Rough
- Turn Finish
- Groove Rough (Only for OD features)
- Groove Finish (Only for OD features)
- Face Rough
- Face Finish

Adding Bar Break moves to the cut passes ensures that the stock edges are deburred and Guide bushing doesn't get damaged when the stock moves in and out of the Guide Bushing during the machining process.

Bar Break Parameters under NC tab of Operation Parameters Dialog Box

Parameters associated with the Bar Breaks functionality has been provided under the ***NC*** tab of the *Operation Parameters* dialog box of all these above operations.



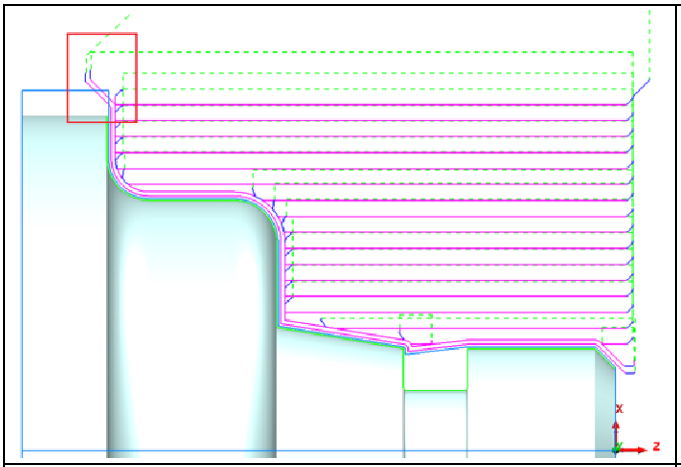
Bar Break Parameters under NC Tab of Operation Parameters Dialog Box for Turn Rough Operation

Bar Break Options

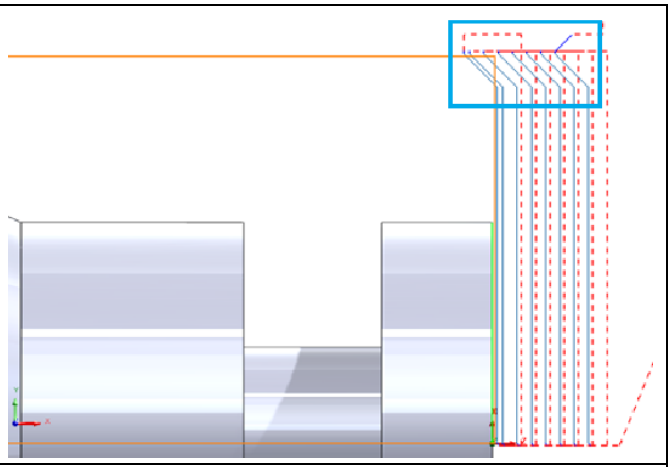
- By default, the ***Bar Break Method*** under the ***NC*** tab will be set to ***None*** i.e., no Bar Break moves.
- Select the ***Chamfer*** option in the *Method* dropdown list to add a chamfer to the sharp edges of the stock along the OD feature. Use the *Distance* and *Angle* parameters within the Bar Break group box to define the chamfer geometry.
- Select the ***Radius*** option in the *Method* dropdown list to add a fillet to the sharp edges of the stock along the OD feature. Use the *Distance* and *Radius* parameters within the Bar Break group box to define the fillet geometry.

Addition of Bar Break Moves to Turning Toolpaths

- For all the supported toolpaths, the Bar Break moves are added to the passes which intersect the max stock diameter. These moves are appended intelligently to the regular cut moves as and when needed.

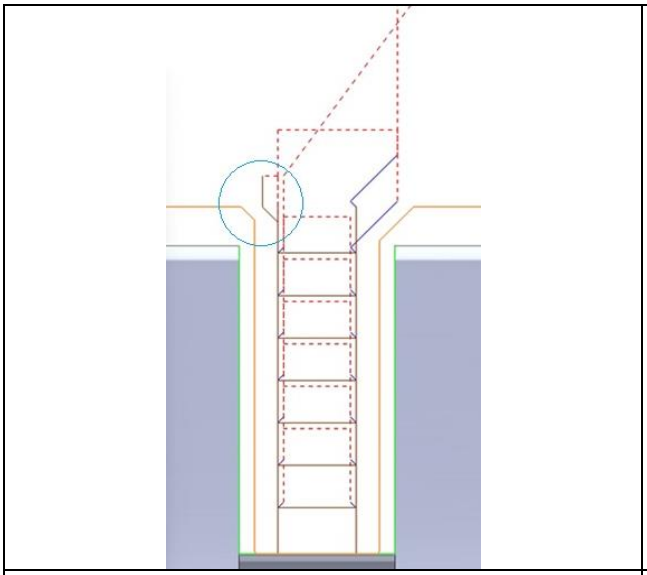


Sample Turn Rough Toolpath with Bar Break Chamfers added to First Cut and Final Cut Passes
(The Bar Break moves are highlighted in a red rectangle.)

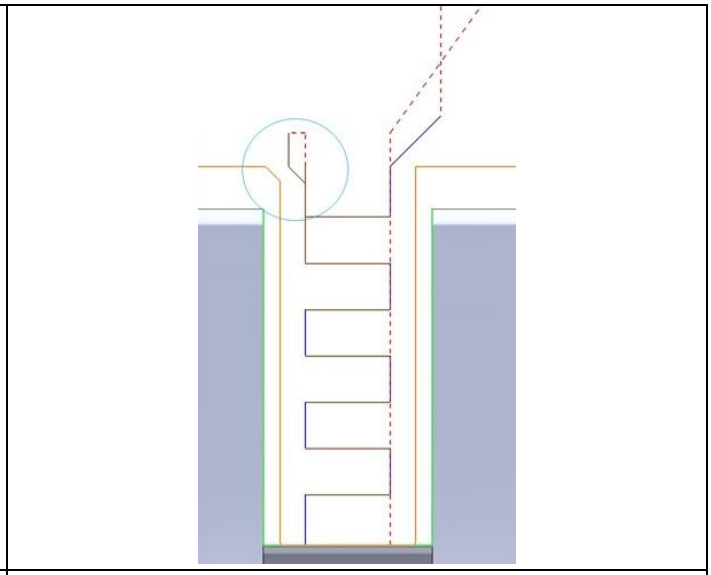


Sample Face Rough Toolpath with Chamfer Bar Break moves for each Cut Pass
(The Bar Break moves are highlighted in the blue rectangle.)

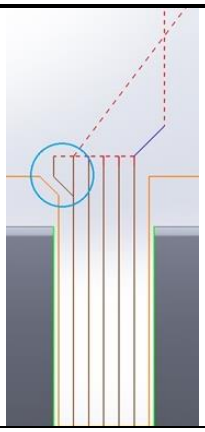
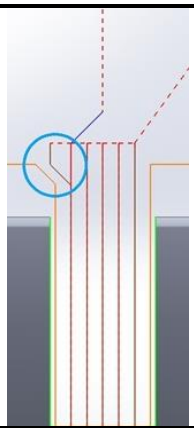
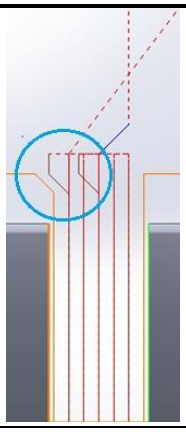
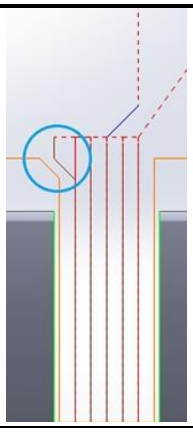
  - For the Groove Rough and Finish toolpaths, the Bar Break moves are added intelligently based on the groove style. The cutting pattern of the groove is considered for adding the bar break moves. There can be only one or multiple Bar Break moves depending on the groove style defined.



Groove Rough Toolpath with 'Zig' Groove Style
(Bar Break Move highlighted in blue circle)



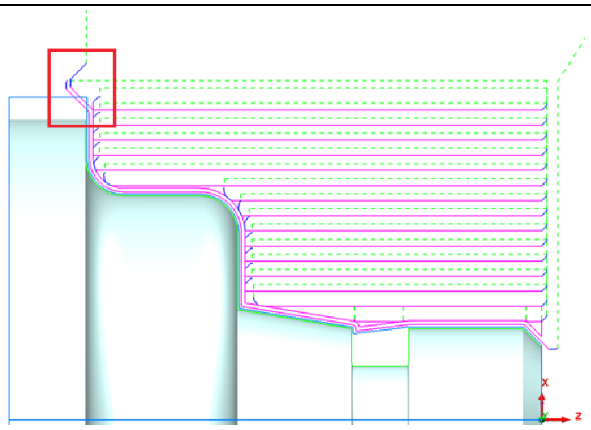
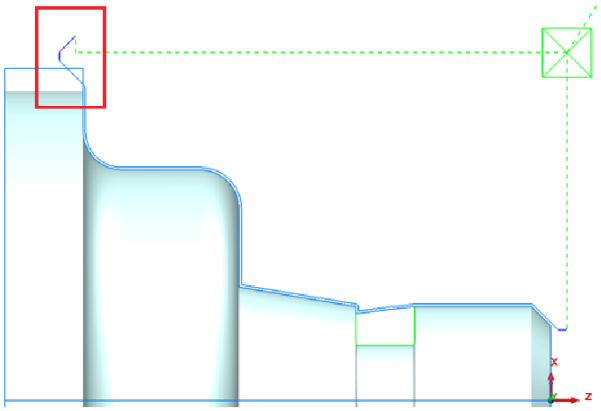
Groove Rough Toolpath with 'Zigzag' Groove Style
(Bar Break Move highlighted in blue circle)

			
Groove Rough Toolpath Order: **S123** Groove Style: Normal Groove Peck Type: None	Groove Rough Toolpath Order: **S321** Groove Style: Normal Groove Peck Type: None	Groove Rough Toolpath Order: **S213** Groove Style: Normal Groove Peck Type: None	Groove Rough Toolpath Order: S231 Groove Style: Normal Groove Peck Type: None
Bar Break move added just before final cut pass.	Bar Break move added just before first cut pass.	Bar Break moves added before first cut and final cut passes.	Bar Break move added just before final cut pass.

Reverse Option for Bar Breaks added to Turn Rough and Turn Finish Toolpaths

Use the ***Reverse*** option within the ***Bar Break*** group box to cut the Bar Break move in a reverse fashion (i.e., the tool will approach the chamfer profile from max stock diameter and will machine it). Placing a check in the *Reverse* checkbox option will machine the Bar Break in the direction opposite to that of the cut passes.

The *Reverse* option is available only for *Turn Rough* and *Turn Finish* operations. However, this checkbox option will be disabled if the *Cut Type* under the *Turn Rough/Turn Finish* tab is reversed.

	
Sample Turn Rough Toolpath with Reversed Cut Type having Bar Break Chamfers added to First Cut and Final Cut Passes	**Sample Turn Finish Toolpath with Reversed Cut Type having Chamfer Bar Break moves all Cut Passes**

Controlling Z Axis Movement for Toolpaths of Parts programmed for Swiss-Turn Machines

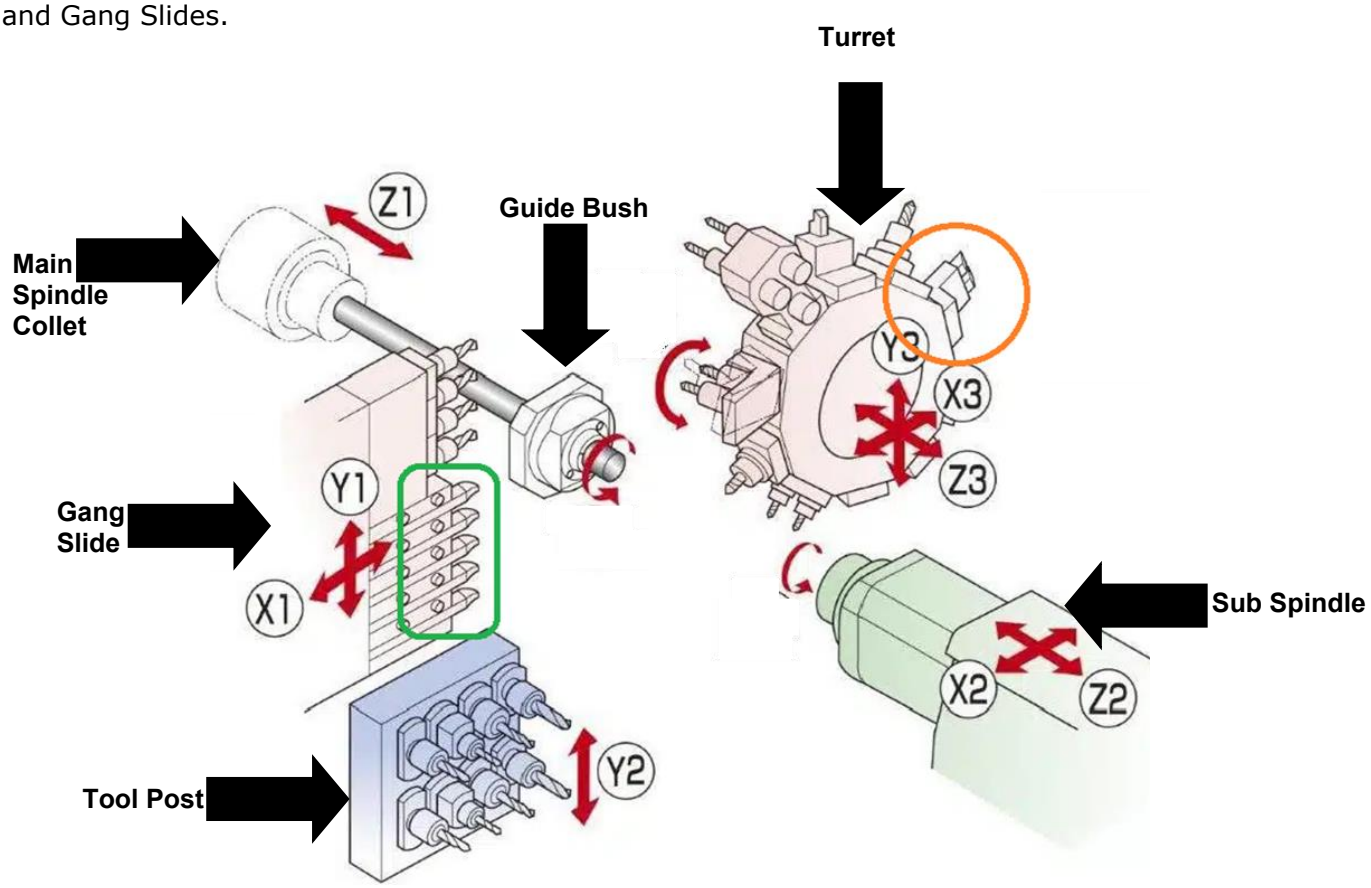
Purpose:

For parts programmed to be machined Swiss Turn/Multi-Function Lathe machines, to assign Z Axis movement to either to the turret or the spindle of the machine depending on whether the tool assigned to an operation is positioned on a turret or Tool Post/Gang Slide

Implementation:

Tool Positioning for Swiss-Turn/Multi-Function Lathe Machines

In most Turn/Mill-Turn machines, the tools are positioned on turrets. In the case of Swiss Turn/Multi-Function Lathe machines, in addition to turrets, the tools can be positioned on Tool Posts and Gang Slides.



Illustrative Image indicating the Tooling Related Components of a Swiss Turn Machine

Z Axis Movement for Toolpaths of Parts programmed for Swiss-Turn/Multi-Function Lathe Machines

Tools positioned on turrets can move in both X and Z axis. In contrast, the tools on some of the tool posts and Gang slide can move only in X or Y axis. For parts programmed to be machined on Swiss-Turn /Multi-Function Lathe machines, the Z axis movement for the toolpaths will be as follows:

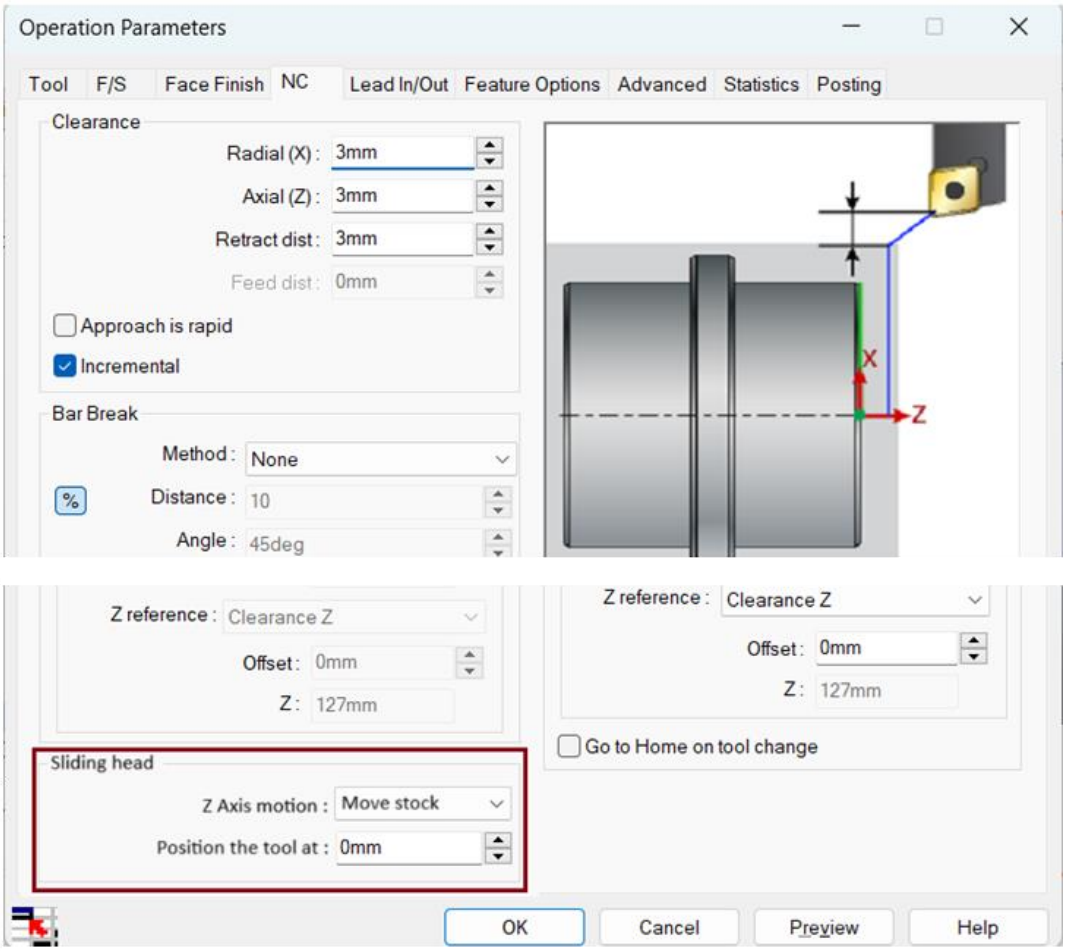
- For toolpaths that have been assigned tools positioned on Tool Posts/Gang Slides, the Z axis movement will be provided by the Main Spindle (i.e., the stock will move in Z axis direction).
- For toolpaths that have been assigned tools positioned on turrets, the Z axis movement will be provided by the turret and the Main/Sub Spindle on which the stock is mounted will remain stationary.

Controlling Z Axis Movement of Toolpaths within CAMWorks

For each machining operation generated, depending on whether the assigned tool is positioned on a turret or Tool Post/Gang Slide, the Z Axis movement needs to be assigned to either to the turret or the spindle of the machine. Within *CAMWorks*, parameters have been introduced within the ***Operation Parameters*** dialog box.

UI Parameters within CAMWorks to Control Z Axis Movement for Toolpaths generated for Swiss Turn Machines

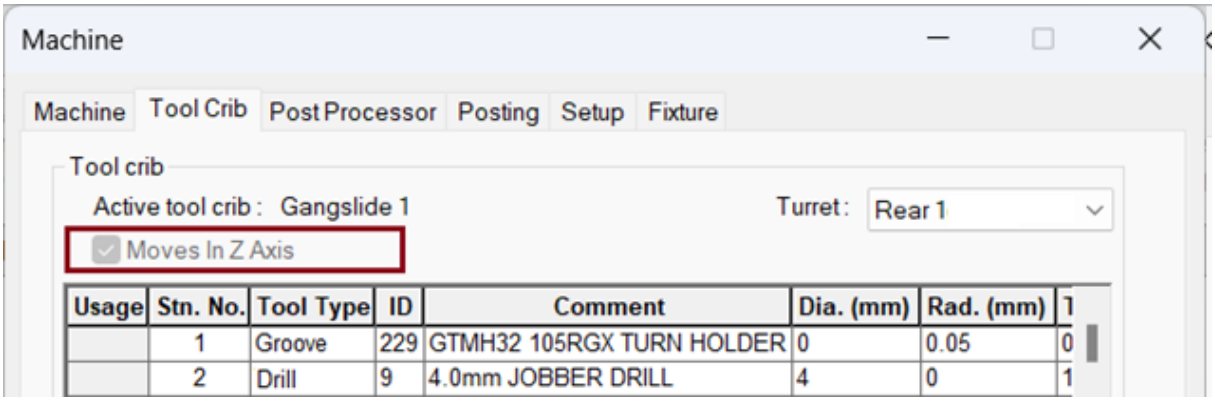
In CAMWorks, for operations that are to be machined on Swiss Turn/Multi-Function Lathe machines with a sliding head, a new group box labelled ***Sliding Head*** has been made available under *NC* tab of the *Operation Parameters* dialog box.



Sliding Head Group Box under NC Tab for Turn Operations

The *Sliding Head* group box and its parameters will be enabled under the *NC* tab of the *Operation Parameters* within the dialog box only if the following conditions are fulfilled:

i. The **Moves in Z Axis** checkbox option under *Tool Crib* tab of the *Machine* dialog box is checked for the toolcrib which is being used to machine the current operation.



'Move in Z Axis' Checkbox Option under Tool Crib Tab of Machine Dialog Box

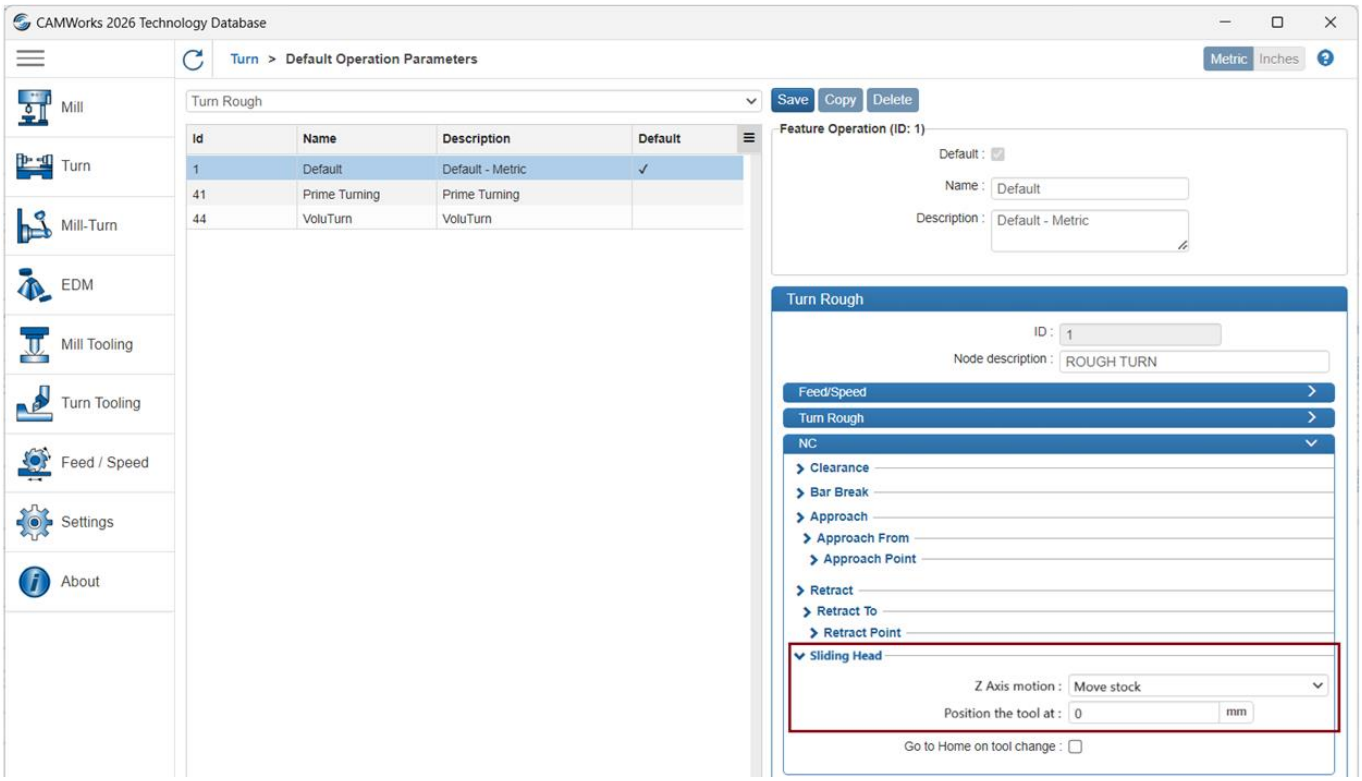
ii. The operation is machined on the Main Spindle.

Parameters within the 'Sliding Head' Group Box under NC Tab of the Operation Parameters Dialog Box

- **Z Axis Motion:** This is a dropdown list containing two options. The default option selected will be based on the option selected for this dropdown list parameter within the TechDB. The following are the options within this dropdown list:
  i. **Move Stock**: When this option is selected, the Z axis movement when machining will be provided by the stock.
  ii. **Move Tool**: When this option is selected, the Z axis movement when machining will be provided by the tool assigned to the operation.
- **Position the tool at:** Use this parameter to indicate the absolute value in Z axis where the tool will be positioned from the face of the Guide Bush. The value you assign must compulsorily be a value greater than zero. The default value for this parameter will be the value defined for this parameter within the TechDB.
- **Position the stock at:** When the Z axis motion type is set to Move Tool then, use this parameter to indicate the position of the stock from the face of the Guide Bush. Based on the value given, the stock will be protruded out of the Guide Bush.

Assigning Default Values within Technology Database for Parameters to Control Z Axis Motion

In TechDB, use the ***NC*** tab under the Turn operations to assign defaults for ***Sliding Head*** parameters.



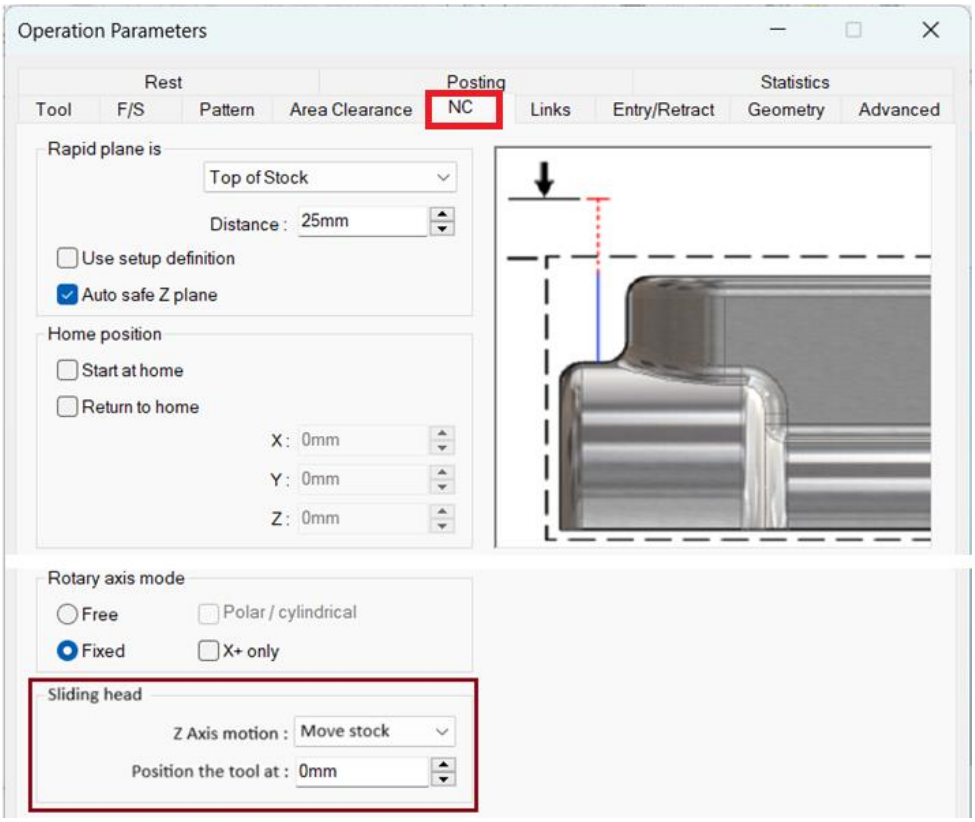
TechDB Form for Assigning Default Operation Parameters for Turn Operations

UI Parameters to Control Z Axis Movement for Turn Toolpaths

For Turn operations, this ***Sliding Head*** group box containing parameters to control the Z axis movement is available under ***NC*** tab of the *Operation Parameters* dialog box.

UI Parameters to Control Z Axis Movement for 2.5 Axis/3 Axis Mill Toolpaths

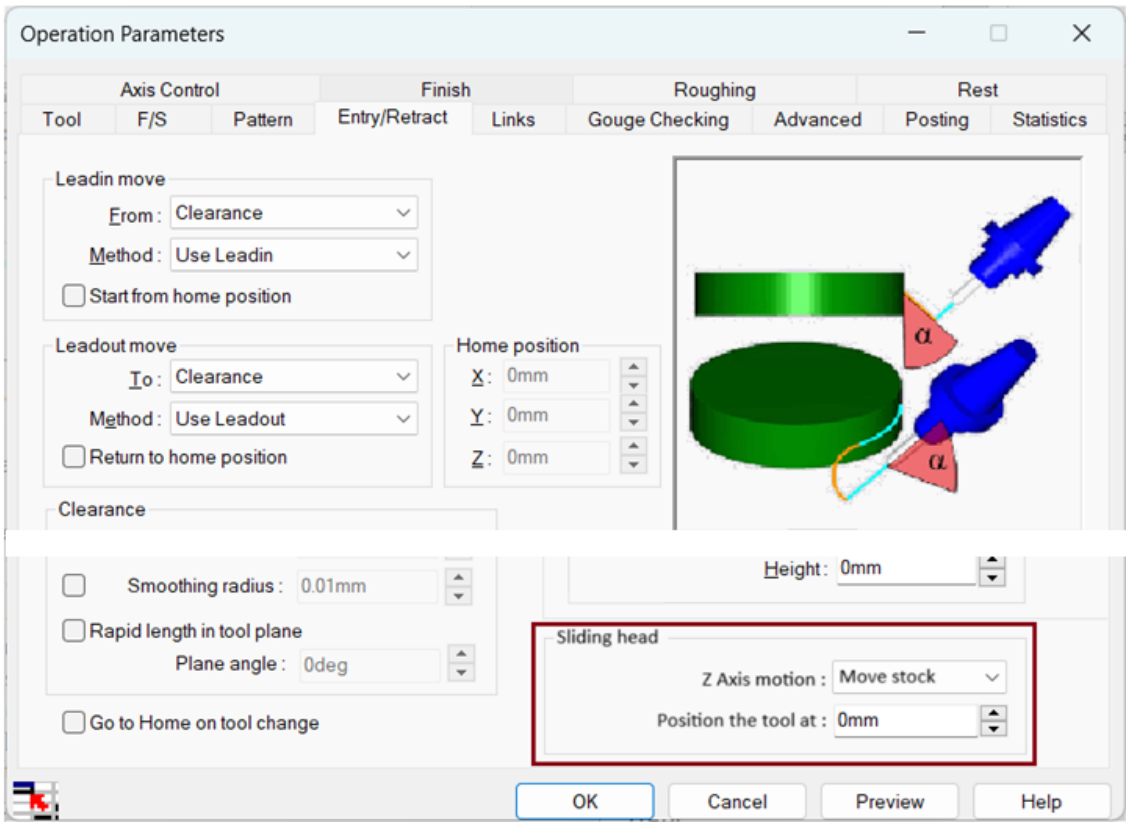
For 2.5 Axis and 3 Axis Mill operations, this ***Sliding Head*** group box containing parameters to control the Z axis movement is available under ***NC*** tab of the *Operation Parameters* dialog box.



Sliding Head Group Box under NC Tab for 3 Axis Mill Operations

UI Parameters to Control Z Axis Movement for Multiaxis Mill Toolpaths

For Multiaxis Mill operations, this ***Sliding Head*** group box containing parameters to control the Z axis movement is available under ***Entry Retract*** tab of the *Operation Parameters* dialog box.



Sliding Head Group Box under Entry/Retract Tab for Multiaxis Mill Operations

Associating Sub Spindle operations with other Operations to ensure Auto-Synchronization

Purpose:

To enable users to associate Sub Spindle operations with any other operation, ensure automatic synchronization of these operations within the Sync Manager, and generate the appropriate sub-spindle support codes in the posted output.

Implementation:

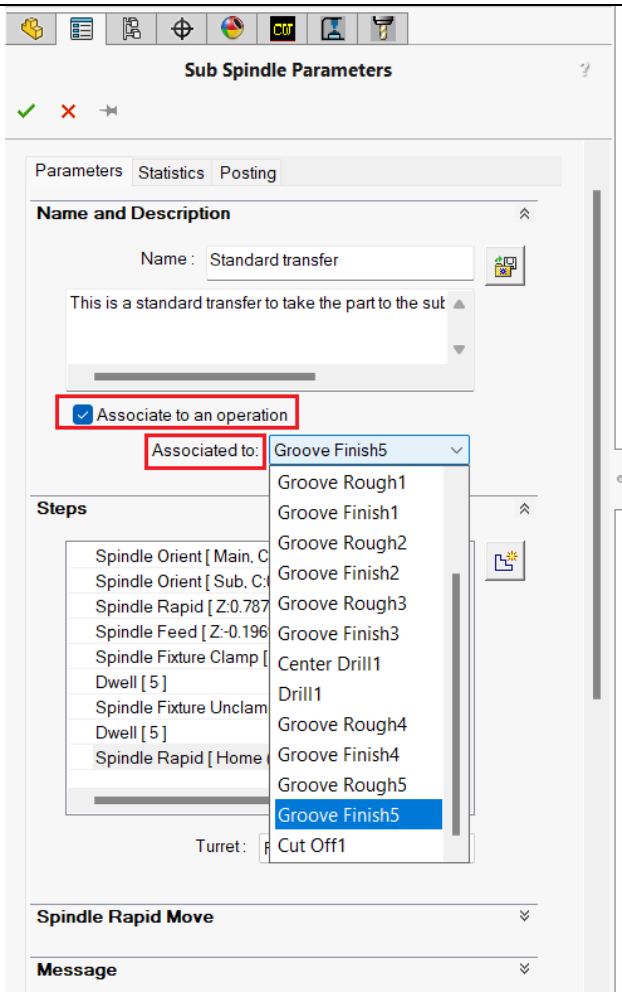
In previous versions of CAMWorks, Sub Spindle operations could not be automatically synchronized with cutting operations.

New Parameters in Sub Spindle Parameters Dialog Box

From *CAMWorks 2026* version onwards, the option to associate Sub spindle operations with other operations has been introduced.
The ***Sub Spindle Parameters*** dialog box is displayed on executing the ***New Sub Spindle Operation*** command.
Under the *Parameters* tab of this dialog box, a new checkbox option ***Associate to an operation*** has been introduced.
Place a check in this checkbox option if you wish to associate the Sub Spindle operation with any of the other operations machined from the Main Spindle. Placing a check in this checkbox will enable the *Associate to* dropdown list. Use this dropdown list to select the operation to associate the Sub Spindle operation with.
(This dropdown list will list all the operations under the selected Turn Setup with the exception of Probe operations, Post operations, and other Sub Spindle operations.)

Advantages of Associating Sub Spindle Operation with other Operations

Associating a Sub Spindle operation with another operation ensures that both these operations are automatically synchronized in the ***CAMWorks Sync Manager***. This functionality is especially useful for synchronizing Cut Off operations with Sub Spindle operations when machining using Turn, Swiss Turn, Mill-Turn or Multi-Function Lathe machines.



New Parameters in the Sub Spindle Parameters UI

Default Selection in 'Associate To' dropdown list:

The *Sub Spindle Operation* dialog box is invoked by executing the *New Sub Spindle Operation* command. The location this command is executed from determines the default selection in the *Associate To* dropdown list.

- If invoked from the *Command Manager*, then the last operation listed under the Turn Setup node in the Operation tree will be listed as the default operation in this dropdown list.
- If invoked from the context menu of the Turn Setup node in the Operation tree, then the last operation listed under that Turn Setup will be listed as the default operation in this dropdown list.
- If invoked from the context menu of any operation node in the Operation tree, then that operation will be listed as the default operation in this dropdown list.

Chip Breaking Functionality for Turn Operations

Purpose:

To provide the option to users to enable the Chip Breaking functionality for Turn operations so that the chips generated during the machining process are short

Implementation:

During the Turn machining process, the machining of metals may result in long chips which can potentially reduce efficiency and tool life. The newly introduced Chip Breaking functionality within the CAMWorks application provides a solution by breaking these chips to smaller, manageable pieces. Enabling the *Chip Breaking* functionality and assigning values to parameters associated with this functionality will modify the resulting toolpath is such a way that the only short chips are formed.

Turn Operations for which Chip Breaking Functionality is Available

The option to enable *Chip Breaking* is available for the following Turn operations:

- Turn Rough operations (within the ***Cut Type*** group box under the ***Turn Rough*** tab)
- Turn Finish operations (within the ***Cut Type*** group box under the ***Turn Finish*** tab)
- Bore Rough operations (within the ***Cut Type*** group box under the ***Bore Rough*** tab)
- Bore Finish operations (within the ***Cut Type*** group box under the ***Bore Finish*** tab)
- Face Rough operations (within the ***Cut Type*** group box under the ***Face Rough*** tab)
- Face Finish operations (within the ***Cut Type*** group box under the ***Face Finish*** tab)

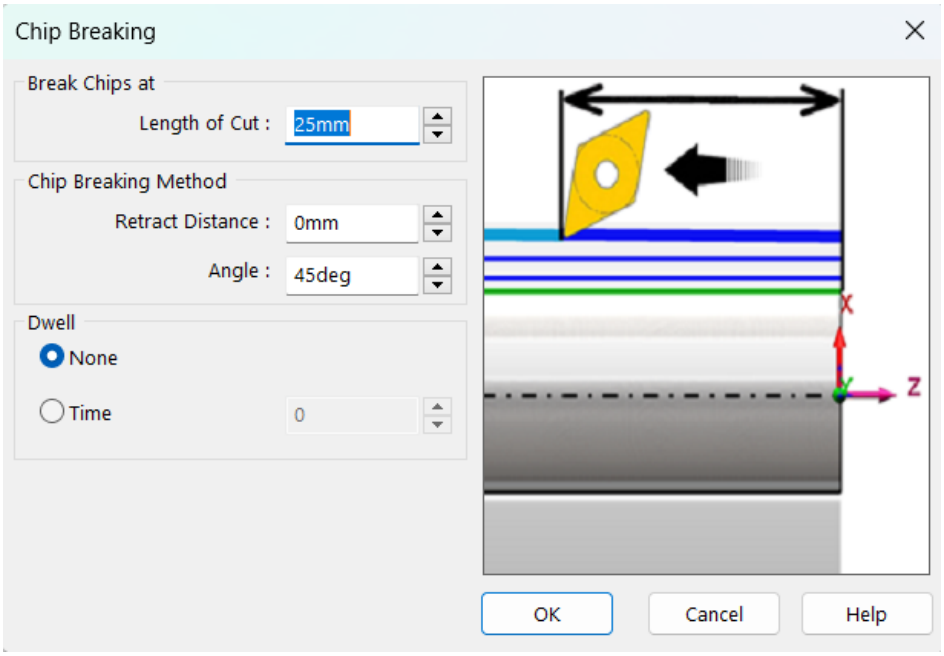
Invoking the Chip Breaking Dialog Box

In the above-listed tab interfaces, within the ***Cut Type*** group box, placing a check in checkbox adjacent to the ***Chip Breaking*** push button will activate ***Chip Breaking*** push button. Clicking on this push button will invoke the ***Chip Breaking*** dialog box.

Note:

- For *Turn Rough*, *Face Rough*, and *Face Rough* operations, the checkbox adjacent to the ***Chip Breaking*** push button will be enabled only if the ***Canned Cycle Output*** checkbox option is unchecked.
- The *Chip Breaking* functionality is not available for *Turn Rough-VoluTurn* and *Turn Rough- Prime Turning* operations.

Parameters in the Chip Breaking Dialog Box



Chip Breaking Dialog Box

Length of Cut:

Use this parameter to indicate the distance along the cut at which a chip breaking point will be induced in the toolpath i.e., the distance between consecutive chip breaking points in the toolpath. The value you assign to this parameter must be greater than zero. The default value for this parameter will be picked from the TechDB. In the default TechDB shipped with the CAMWorks application, this value will be 25mm (1 inch).

Retract Distance

Use this parameter to indicate the distance the tool must traverse in the reverse direction from the chip break point. The value can be 0 or more. The default value will be zero or as set in the TechDB.

Angle

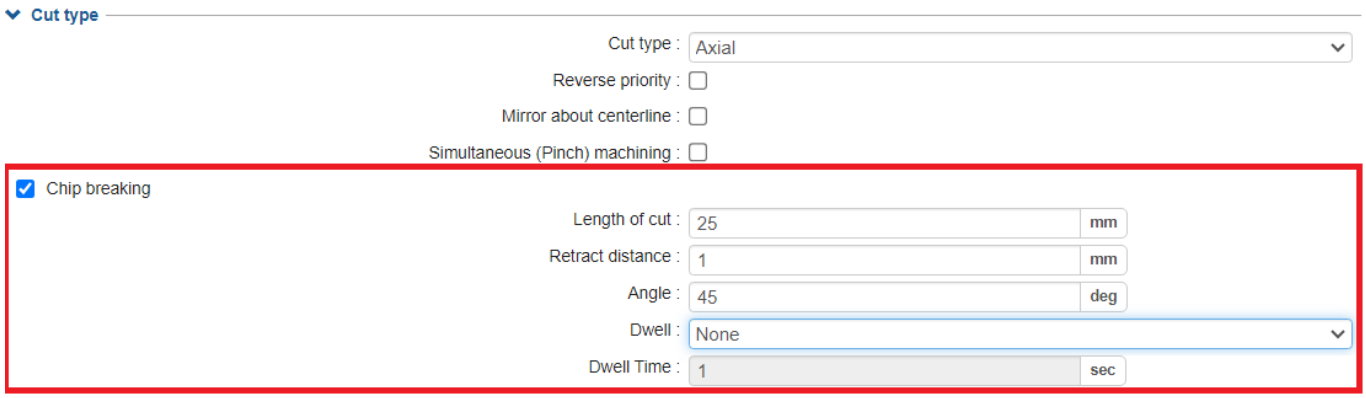
When Chip breaking is enabled for a toolpath, the tool traverses in the reverse direction from the chip break point based on the value assigned to the Retract distance parameter. For this retract move, the angle will be measured with respect to the segment of the toolpath in which the Chip breaking point is located. The default value will be 45 degrees or as set in the TechDB. The allowed value range for this parameter is from 0 degrees to 90 degrees.

Dwell

- When the *None* option is selected for this parameter, the tool will not dwell after traversing the retract distance. It will immediately restart the cutting moves as per the toolpath.
- When the *Time* option is selected, the tool will dwell after traversing the retract distance. Use the parametric field adjacent to this option to indicate the dwell time in seconds. The tool will remain at the end of the retract distance for the time duration mentioned in this field. The default value for this parameter will be picked from the TechDB.

Assigning Default Values for Parameters associated with Chip Breaking functionality in TechDB

In the corresponding ***Operation*** Forms for supported Turn operations, placing a check in the ***Chip Breaking*** checkbox will activate parameters associated with the Chip Breaking functionality. Use these parameters to assign/edit these parameters.



Chip Breaking Parameters in Technology Database

Simulation and Visualization

Display of Machine Kinematics during Toolpath Simulation

Purpose:

Support for displaying Machine Kinematics during Toolpath Simulation

Implementation:

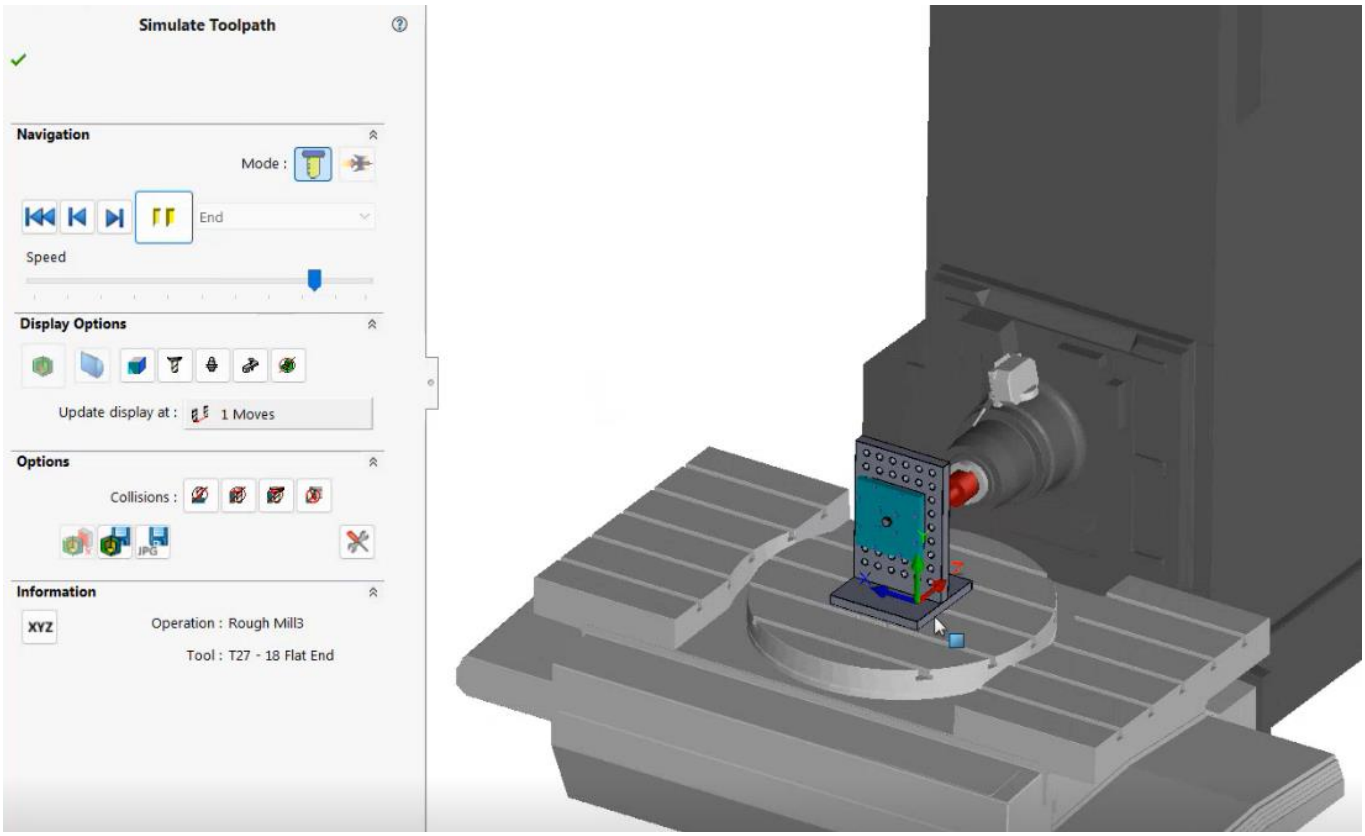
In the graphics area of CAMWorks, the commands available within the *Display Components* toolbar are used to enable/disable the display of the various groups of machine components comprising the Simulation Machine.

From *CAMWorks 2026* version onwards, if the commands available within the *Display Components* toolbar are enabled to display machine components, then on executing the *Simulate Toolpath* command, Machine Kinematics will be observed in the graphics area during the entire simulation process.

Mill Mode

In Mill mode, Machine Kinematics simulation is supported for the following Mill Machine types:

- 3 Axis Mill Machines
- 4 Axis Mill Machines
- Horizontal 4 Axis Mill Machines

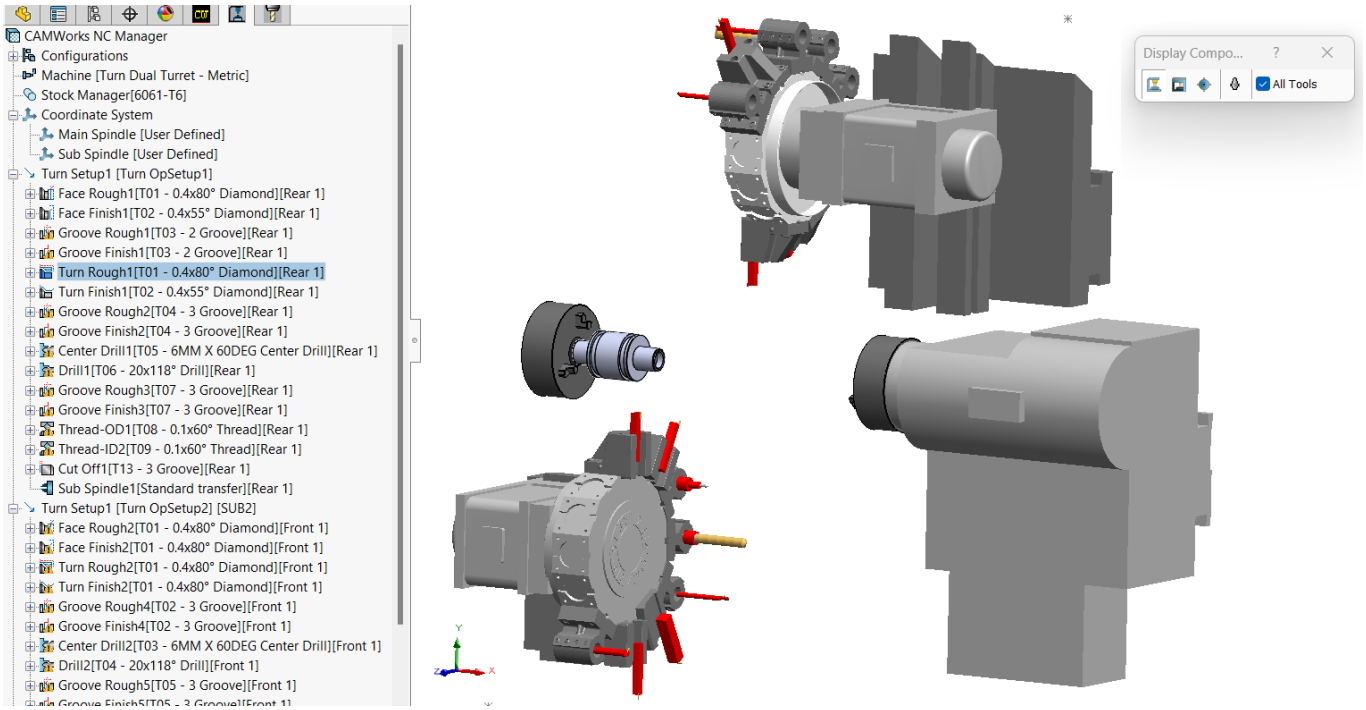


Machine Kinematics in Graphics Area during Toolpath Simulation for a Horizontal 4 Axis Mill Machine

Turn Mode

In Turn mode, Machine Kinematics simulation is supported for the following Turn Machine types:

- Single Turret Turn Machines (2 Axis Turn)
- Dual Turret Turn Machines (4 Axis Turn)
- Multi Turret Turn Machines

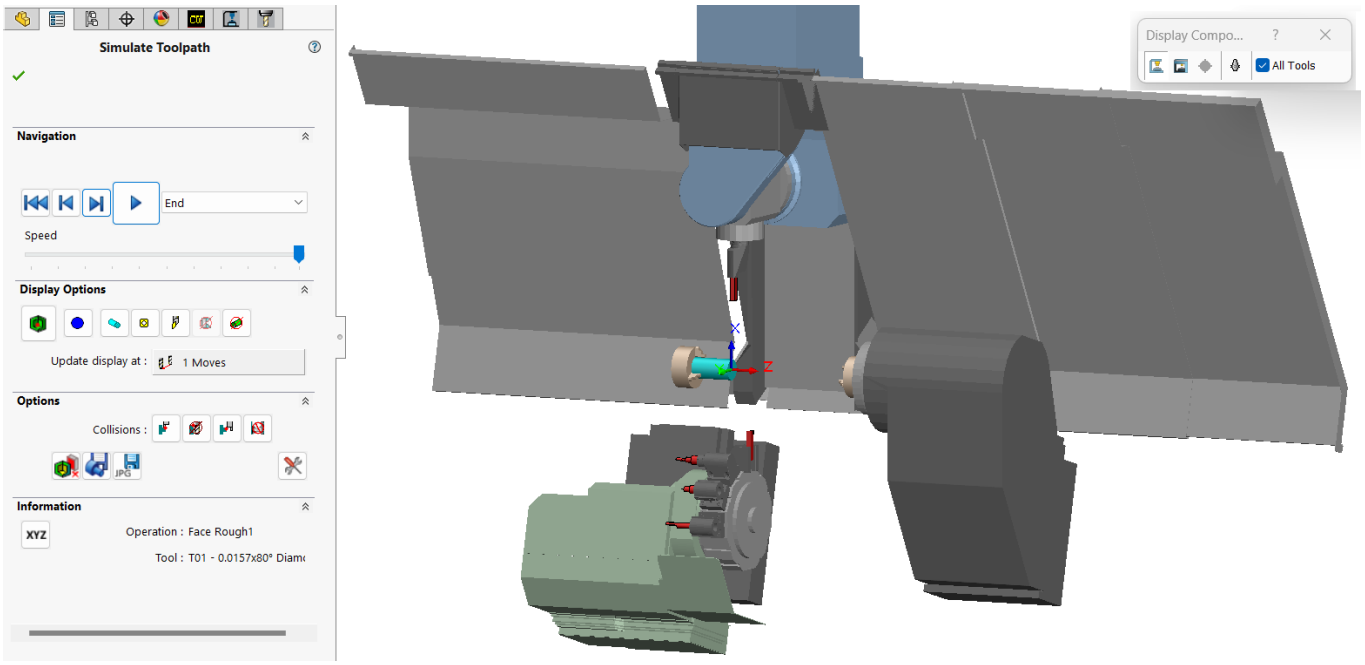


Machine Kinematics in Graphics Area during Toolpath Simulation for a Turn Machine

Mill-Turn Mode

In Mill-Turn mode, Machine Kinematics simulation is supported for the following Machine types:

- Single Turret Mill-Turn Machines
- Dual Turret Mill-Turn Machines
- Multi Turret Mill-Turn Machines
- 1H_1T Mill-Turn Machines (with one Turret and one Mill tool crib)



Machine Kinematics in Graphics Area during Toolpath Simulation for a 1H 1T Mill-Turn Machine

Machine Awareness Functionality Support For CNC Machines (Including Swiss Lathes)

Purpose:

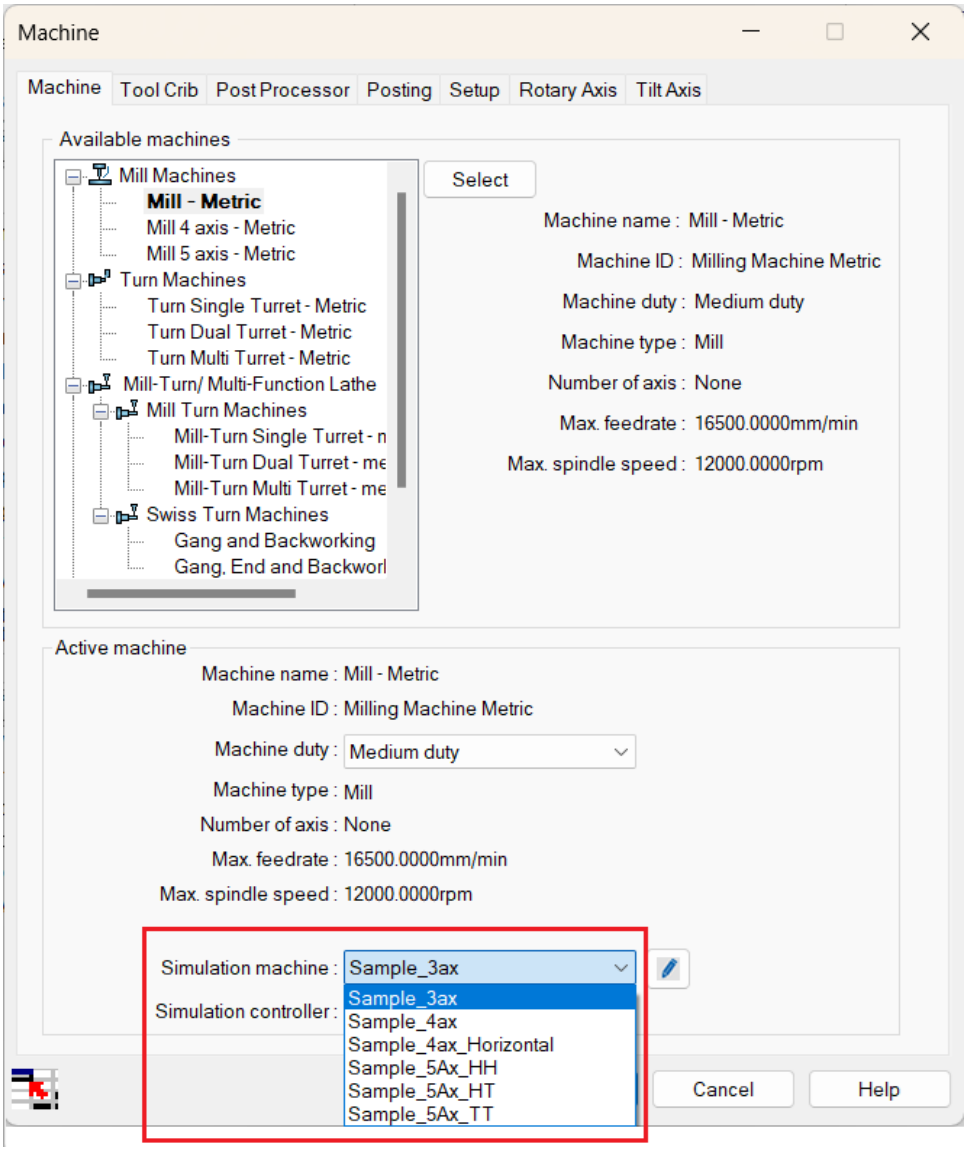
To ensure *Machine Awareness* functionality is supported for all CNC machines within the CAMWorks application

Implementation:

Machine Awareness refers to *CAMWorks'* ability to interpret, display, replicate and simulate the physical environment of the selected CNC machine in the graphics area. *CAMWorks* achieves Machine Awareness by creating a digital representation of the selected CNC machine, inclusive of all its components, to allow for a realistic simulation.

Pre-requisite for Enabling Machine Awareness Functionality

- To access the *Machine Awareness* functionality in *CAMWorks*, your *CAMWorks* license must be configured to run this functionality. Contact your reseller if you wish to get this functionality activated.
- Once your license is configured to execute the *Machine Awareness* functionality, the ***Simulation Machine*** dropdown list under ***Machine*** tab of the ***Machine*** dialog box will display a list of all the virtual simulation machines that can be simulated. Use this dropdown list to select the machine to be simulated. Selecting a *Simulation Machine* in the *Simulation Machine* dropdown list is a pre-requisite to enable Machine Awareness for the specific part/assembly being machined.



Simulation Machine Dropdown list under Machine tab of Machine Dialog Box

Advantages of Using 'Machine Awareness' Functionality

- **Graphical Awareness:** Machine awareness provides CAM programmers the ability to create a CNC setup and program a part or assembly on a digital twin of the CNC machine. As *CAMWorks* is seamlessly integrated with *SOLIDWORKS/CAMWorks Solids*, this digital twin of the selected CNC machine can be viewed in the graphics area along with the part/assembly being machined. By including a digital twin of the CNC machine during the programming workflow, CAM programmers are provided with instant graphic awareness of the machine's table size, travel limits, rotational limits, and available tooling. This helps in ensuring the viability of the planned machining process.
- **Improved Toolpath verification:** Machine awareness helps in understanding the selected CNC machine's capabilities and limitations. This helps programmers to generate efficient and reliable toolpaths.

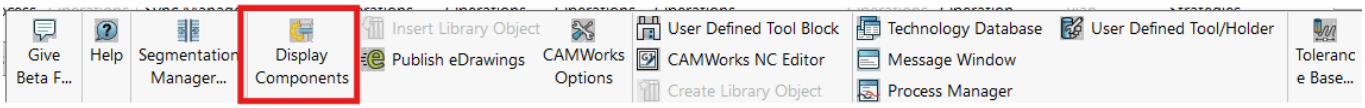
From *CAMWorks 2026* version onwards, the *Machine Awareness* functionality has been further enhanced to ensure that the display and simulation of the machine components is rendered more realistically vis-à-vis the previous versions.

Activating 'Machine Awareness' using the 'Display Components' Toolbar

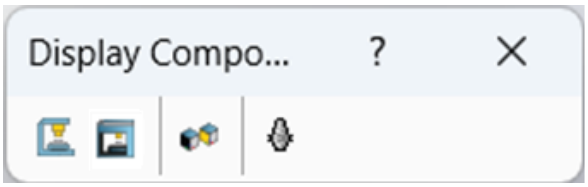
The enabling/disabling of the display of various machine components (i.e., *Machine Awareness*) is controlled through the ***Display Components*** toolbar.

The ***Display Components*** toolbar will be displayed in the graphics area of the *CAMWorks* user interface in the following scenarios:

- When you execute the ***Step Through Toolpath*** command
- When you execute the ***Simulate Toolpath*** command
- When you click on the ***Display Components*** command available within the ***CAMWorks Command Manager***



'Display Components' Command in CAMWorks Command Manager



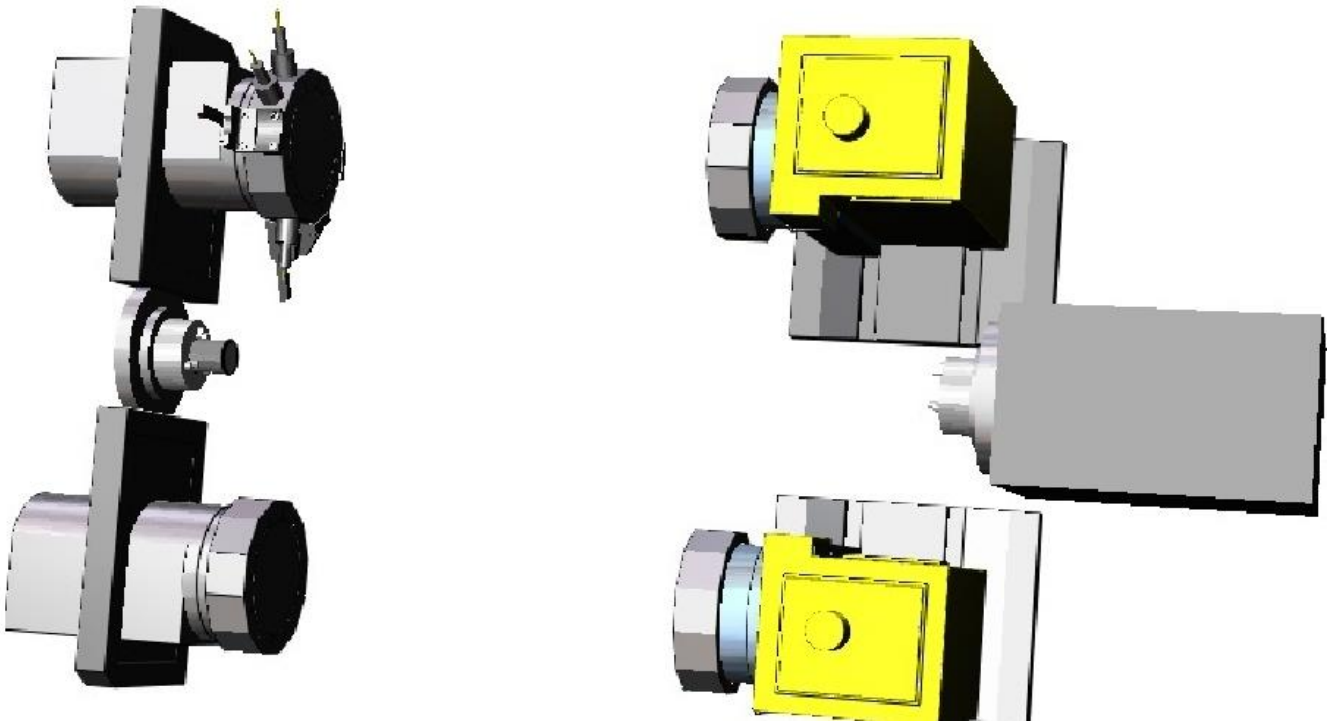
Display Machine Components Toolbar as displayed in the Graphics Area

Display Component toolbar Commands enabled During Toolpath Simulation

The following command buttons and options of the ***Display Components*** toolbar will be enabled during the Toolpath Simulation process:

- **Display Machine:** Use this toggle button to enable/disable the display of machine components of the Simulation Machine in the graphics area.
- **Display Housing:** Use this toggle button to enable/disable the display of the 'machine housing' in which the machine components are installed. This toggle button will be disabled in the toolbar during execution of the ***Step Through*** toolpath and ***Simulate Toolpath*** commands.)
- **Virtual Machine Stock Mounting:** This button will be available in the ***Display Components*** toolbar only for Mill machines. This button will be disabled in the toolbar during execution of the ***Step Through toolpath*** and ***Simulate Toolpath*** commands.) When this toggle button is set to 'ON' status, the dialog box to define the virtual machine stock mounting point will be displayed.

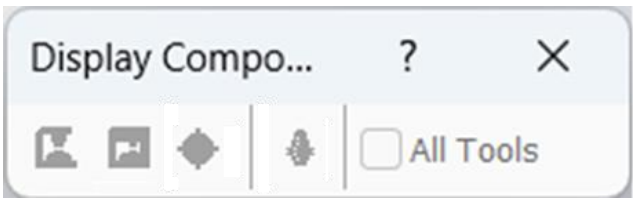
- **Display Turret:** This toggle button is displayed within the toolbar only when the selected Simulation machine is a Turn or Mill-Turn/Multi-Function Lathe machine. Use this toggle button to enable/disable the display of the turret(s) and tools loaded on those turrets.
- **Tool & Tool Holder Display:** Use this button to change the display of the machining tools and holders in the graphics area.
- **All Tools (Checkbox):** This checkbox is displayed within the toolbar only when the selected Simulation machine is a Turn or Mill-Turn/Multi-Function Lathe machine. Use this checkbox option to enable/ disable the display of tools loaded on the turret in the graphics area.



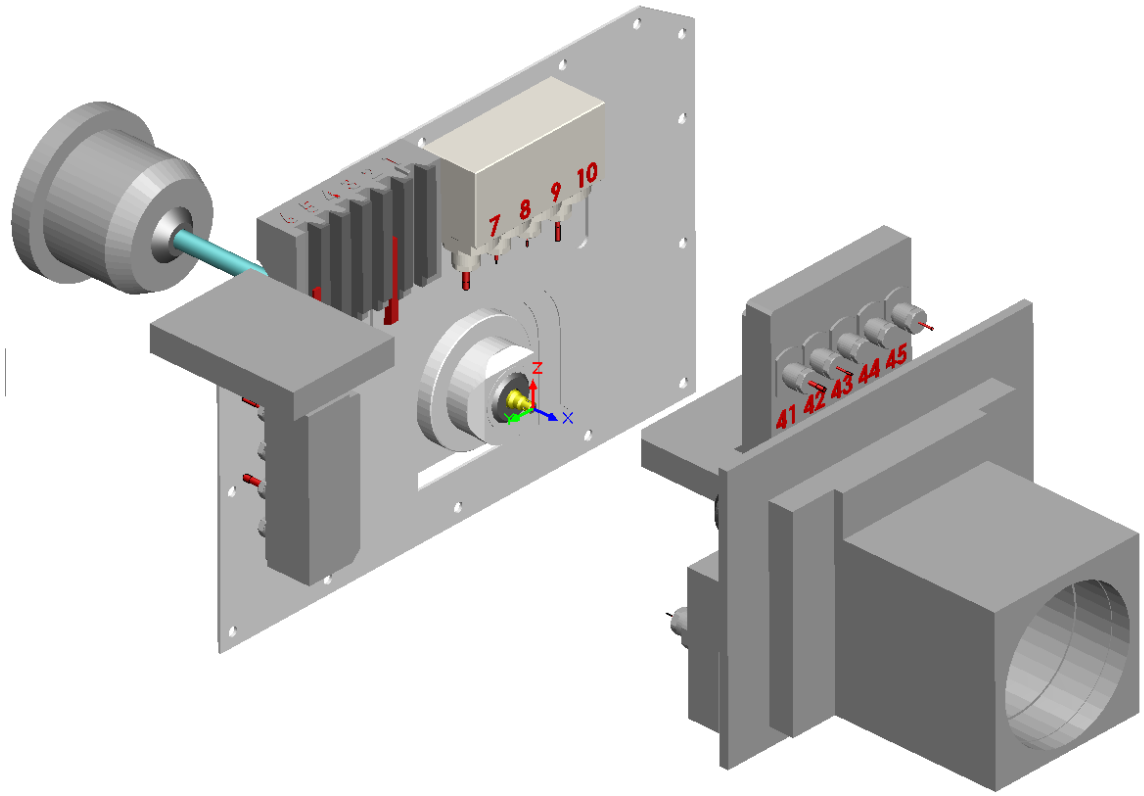
Sample Toolpath Simulation when Simulation Machine is a Multi-Function Lathe Machine

Machine Awareness Functionality for Mill-Turn/ Multi-Function Lathes/ Swiss Lathes

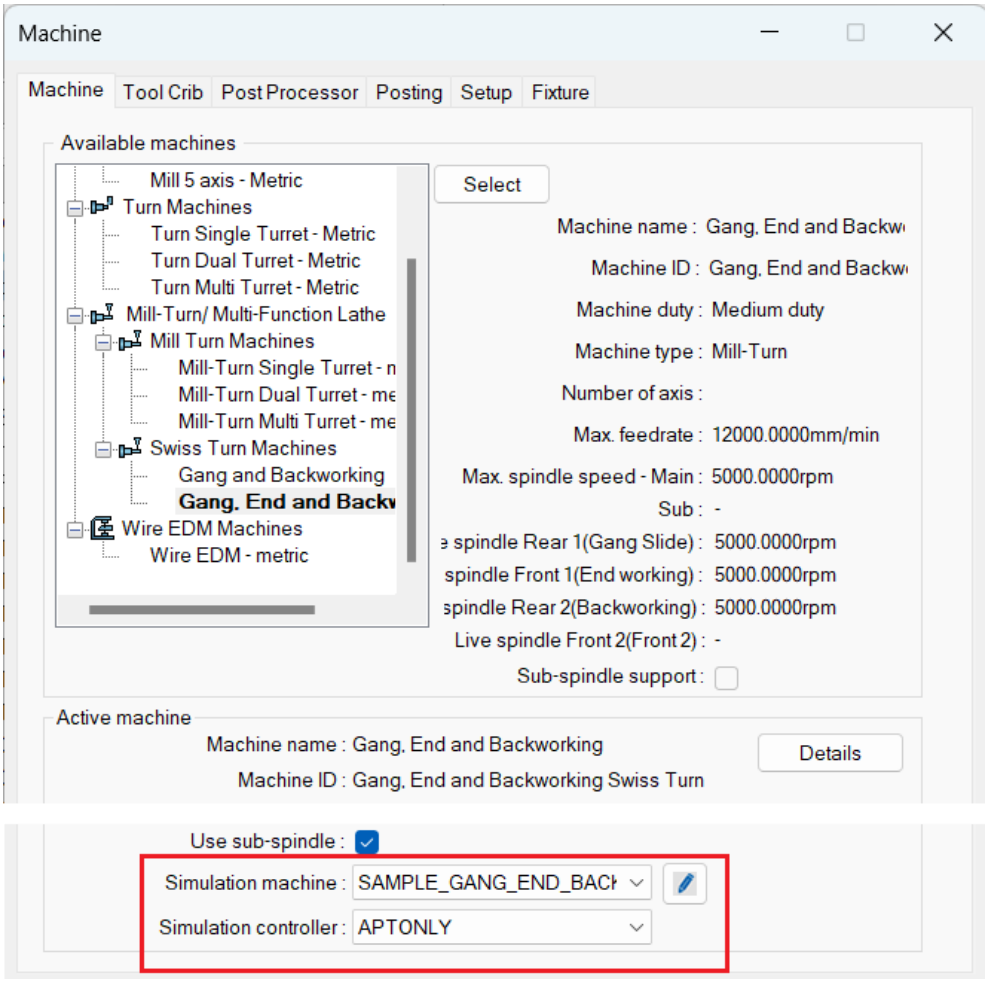
The *Machine Awareness* functionality will be supported for the newly introduced Swiss Turn machines. To render Machine Awareness for a Swiss Turn machine, ensure that the *Simulation Machine* template file and *Controller emulation* template file are selected under *Machine* tab of the *Machine* dialog box when programming the part/assembly.
Note that *Machine Awareness* functionality is not supported for ***EWC (Encrypted Work Cell)*** based machines (i.e., encrypted machines). If the *Simulation Machine* selected is such an encrypted machine, then all commands within the ***Display Components*** toolbar will be disabled.



Display Machine Components Toolbar with all Commands Disabled



Illustrative Image displaying Machine Awareness for a Swiss Turn Machine



Select the desired Swiss Turn Template and Controller under Machine tab of Machine Dialog Box

Support for Swiss Turn Machines in CAMWorks Virtual Machine Environment

Purpose:

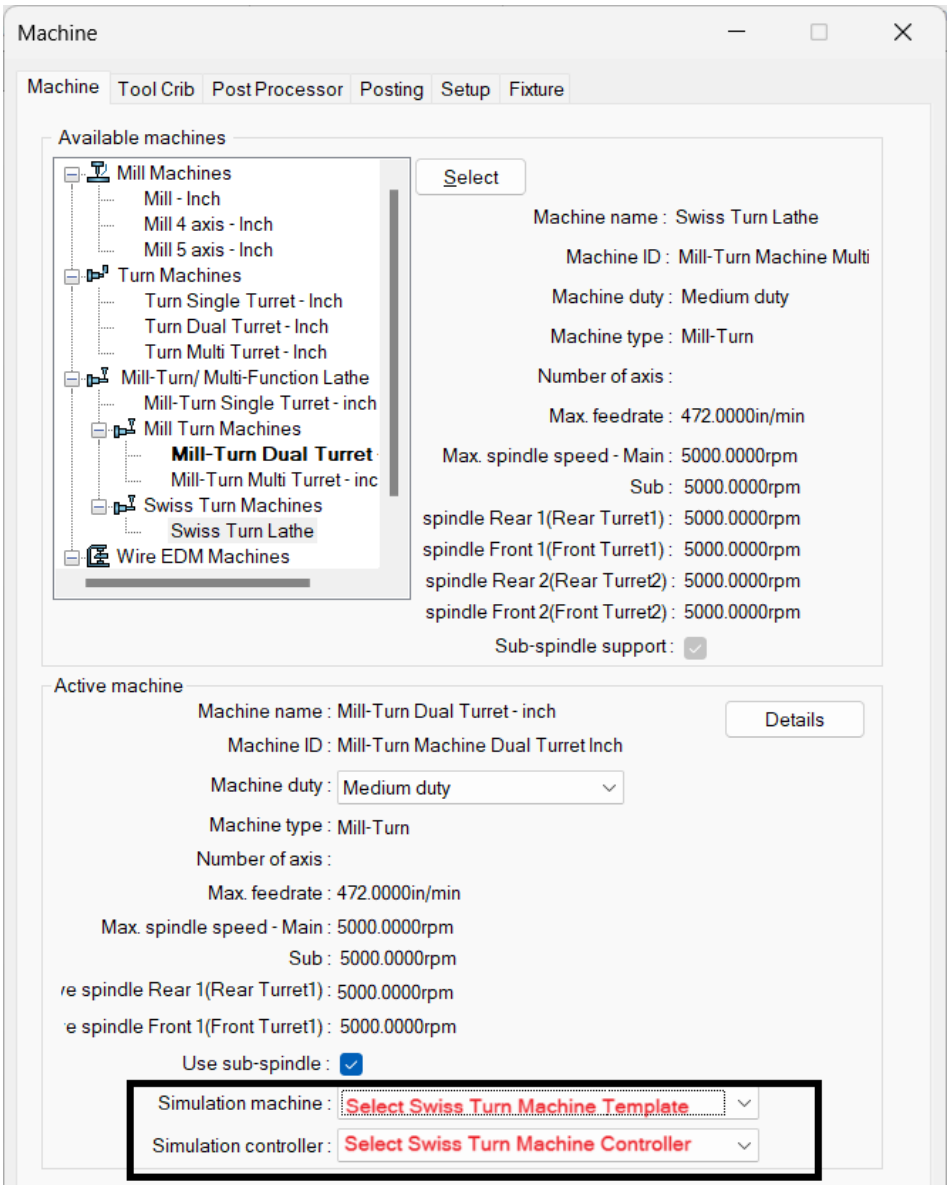
Support for simulation of programmed parts/assemblies to be machined using Swiss Turn machines within the *CAMWorks Virtual Machine* application

Implementation:

The *CAMWorks Virtual Machine* is the resident Machine Simulation application available as part of the CAMWorks suite of licensed modules. G-codes generated for parts/assemblies programmed using the CAMWorks application can be simulated in the *CAMWorks Virtual Machine* application. To use this application, your CAMWorks license must be configured to run this application.

From *CAMWorks 2026* version onwards, parts/assemblies can be programmed to be machined on Swiss Turn machines. The G-code generated for such parts/assemblies can be simulated in the *CAMWorks Virtual Machine* application.

The *CAMWorks Virtual Machine* application will have a pre-defined library of Machine templates and CNC controllers for all demo Swiss Turn machines shipped with the CAMWorks application. If you wish to simulate a custom Swiss Turn machine within the *CAMWorks Virtual Machine* environment, then ensure that you procure its corresponding *Simulation Machine* template file and *Controller emulation* template file from your reseller. Ensure that these template files are selected under *Machine* tab of the *Machine* dialog box when programming the part/assembly.



Select Desired Simulation Machine and Controller under Machine Tab of Machine Dialog box

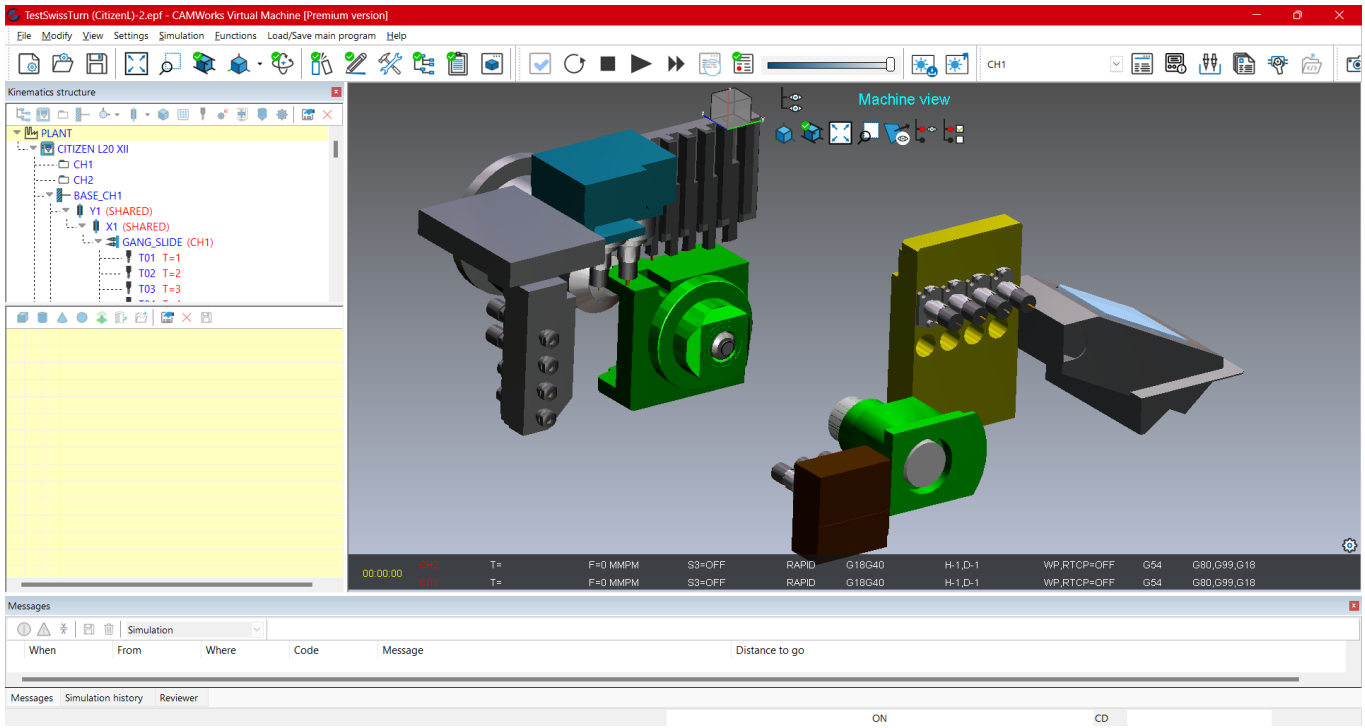


Image Demonstrating Swiss Turn Machine Simulation within the CAMWorks Virtual Machine Environment

Expanded Sync Manager View for CAMWorks Sync Manager

Purpose:

To provide the option for an expanded view of synchronized operations machined on different spindles (Main Spindle/ Sub Spindle) using tools from different turrets/tool posts within the ***CAMWorks Sync Manager*** dialog box

Implementation:

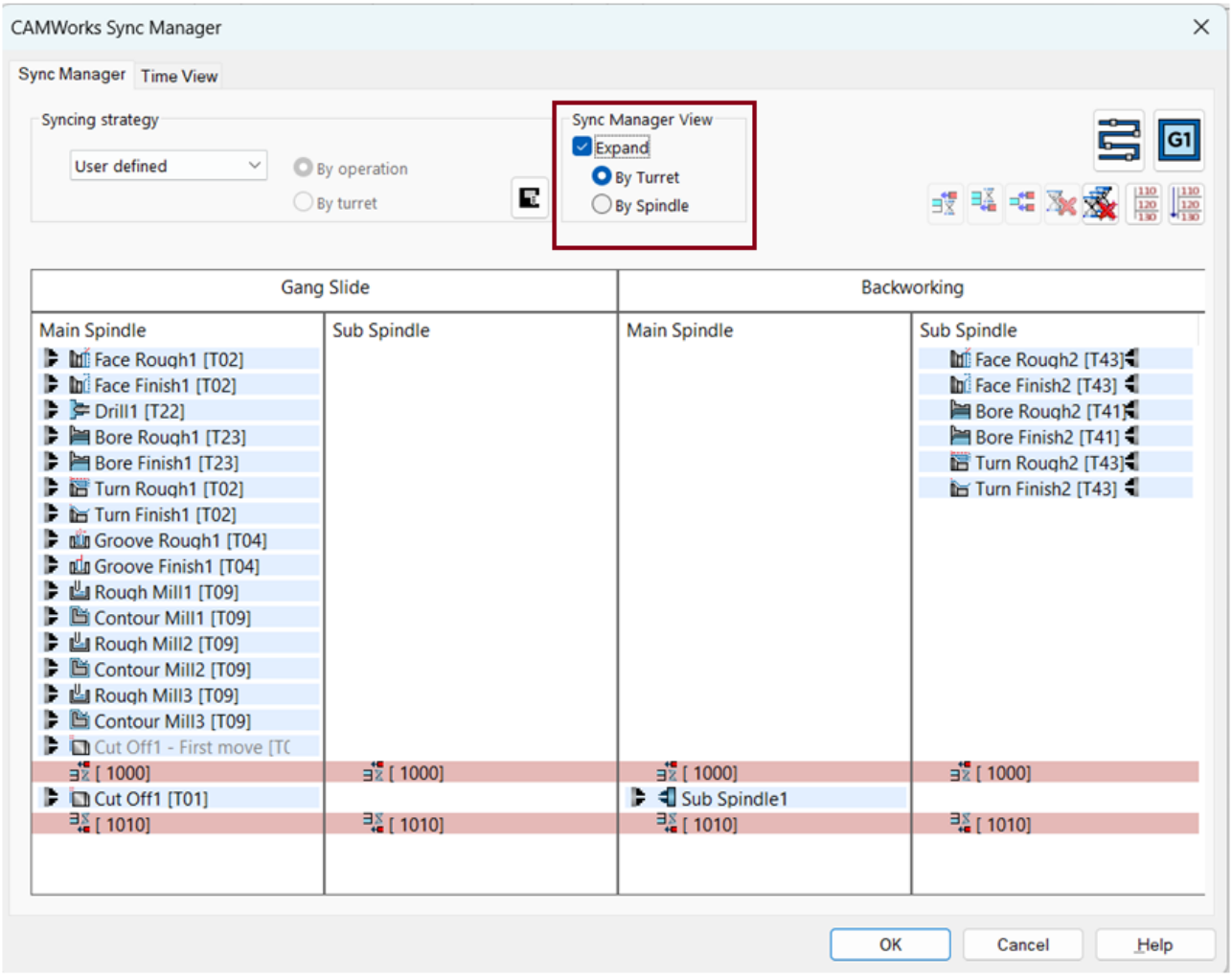
In previous versions of CAMWorks, for Turn and Mill-Turn machines, the ***CAMWorks Sync Manager*** functionality allowed you to synchronize the toolpaths machined by the tools in two or more turrets/tool cribs/tool posts. This was achieved by re-ordering the operations in the ***CAMWorks Sync Manager*** user interface. Optimal synchronization can drastically reduce the time required to machine the parts/assemblies.

From *CAMWorks 2026* version onwards, the ***CAMWorks Sync Manager*** functionality has been enhanced to provide an expanded view of the synchronized operations. If the ***Syncing Strategy*** is set to ***User Defined*** and the ***Expand*** checkbox within the ***Sync Manager View*** group box is checked with the ***By Turret*** or ***By Spindle*** option selected, then an expanded view of the ***Sync Manager grid*** based on the Turret-Spindle or Spindle-Turret combinations will be displayed.

'By Turret' Option for Expanded Sync Manager View

If the ***By Turret*** view is selected, the ***Sync Manager grid*** will have a column header indicating the Turret info.

- The Turret column will be sub divided for the spindles available on the machine. Viz. Main and Sub spindle.
- Each sub column will list the operations having the tools assigned from the selected Turret or Tool post.

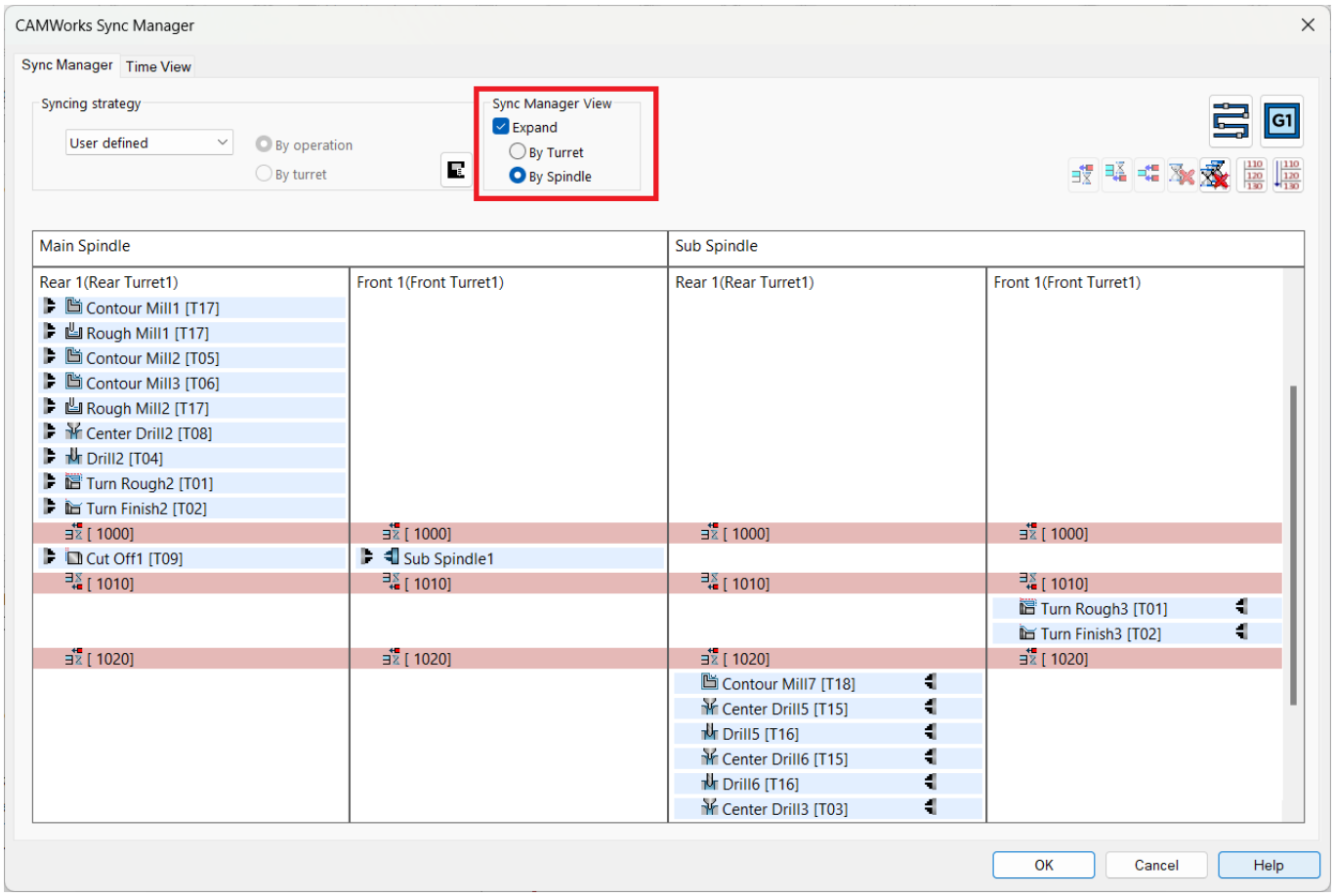


Expanded Sync Manager View when 'By Turret' Option is Selected

'By Spindle' Option for Expanded Sync Manager View

If the ***By Spindle*** option is selected, the ***Sync Manager grid*** will have two-tier column headers that indicate the Turret-Spindle combination info.

- The first-level column header will be the name of the Spindle (viz. 'Main Spindle' and 'Sub Spindle').
- Each first-level header indicating the Spindle will span across one or more second-level column headers that indicate the name of the turret.
- Each column in the grid will list only the operations that are machined from that specific spindle (first-level column header) and have tools assigned from that specific turret (second-level column header). These operations will be listed as per their machining sequence within each column.



Expanded Sync Manager View when 'By Spindle' Option is Selected

Providing Visual Feedback on Start Location of Wait Codes within CAMWorks Sync Manager

Purpose:

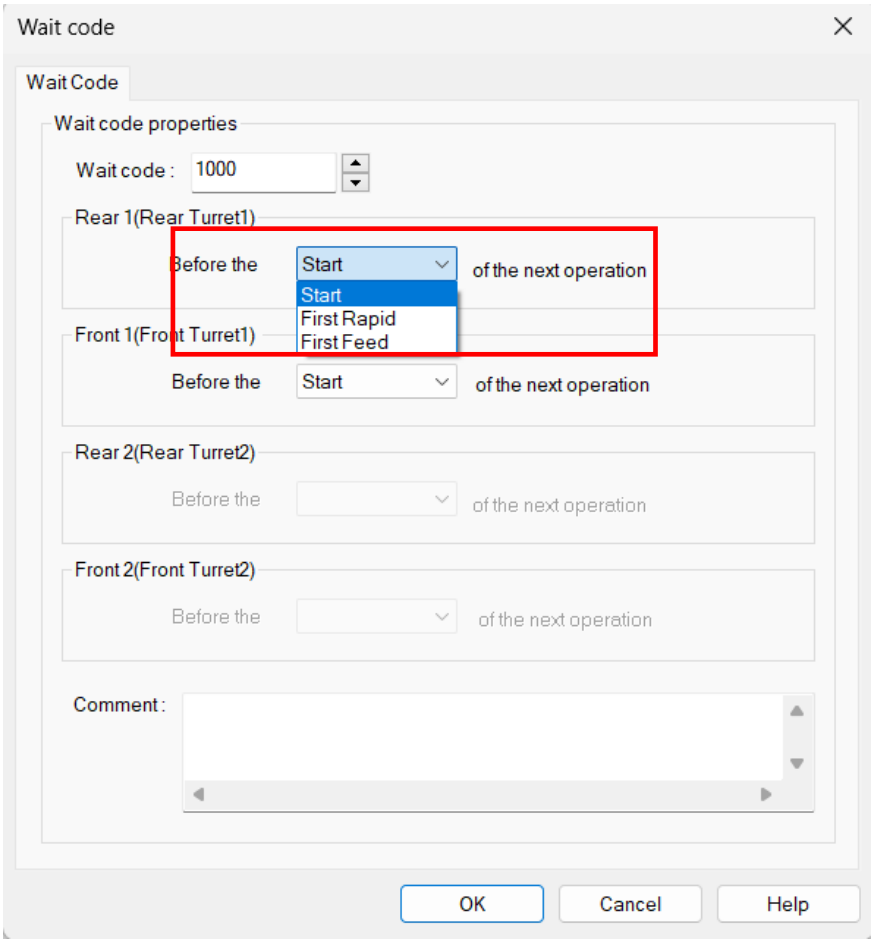
To provide the visual feedback about the start location of the wait codes for synchronized operations within the Sync Manager grid of the ***CAMWorks Sync Manager*** dialog box to indicate whether the wait codes is positioned after a tool change or after the first feed move

Implementation:

Within the ***Sync Manager*** tab under the ***CAMWorks Sync Manager*** dialog box, wait codes can be added to operations machined from different turrets/tool posts/tool cribs to sync them with other operations. Using the ***Wait Code*** commands available within this tab, wait codes can be added at the following locations:

- Before the start of the next operation 
- Before the first rapid move of the next operation 
- Before the first feed move of the next operation 

For a wait code already inserted, its start location can be edited by double-clicking on the wait code to invoke the *Wait Code* dialog box.



Wait Code Properties Dialog Box

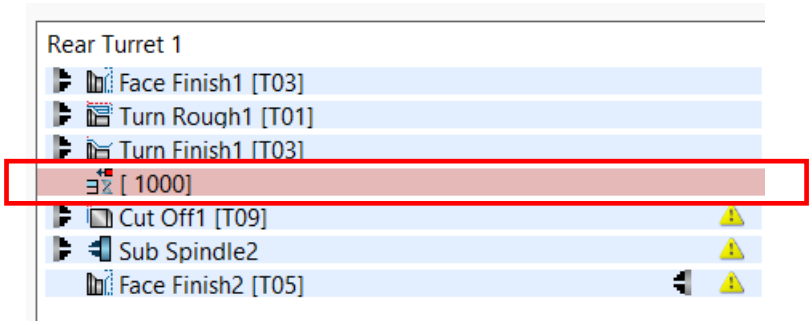
In previous versions of CAMWorks, all inserted wait codes were visible within the ***Sync Manager*** grid under ***Sync Manager*** tab. However, there was no visual info on their start locations i.e., whether the wait code was inserted before the start of the operation or before the first rapid move or feed move of the operation.

From *CAMWorks 2026* version onwards, a mechanism to provide visual indication about the start location of the inserted wait codes within the ***Sync Manager*** grid under ***Sync Manager*** tab has been introduced. This visual feedback is provided in the form of nodes that indicate the location of the wait code.

Start Location Indicator for Wait Code when Wait Code Property is set to 'Before the Start of the next operation'

If the wait code property indicating the start position is set to ***'Before the Start of the next operation'***, then no entry indicating the start location of that wait code will be displayed in the *Sync Manager* grid under the *Sync Manager* tab.

The wait code will be located immediately above the operation in the column of the *Sync Manager* grid.



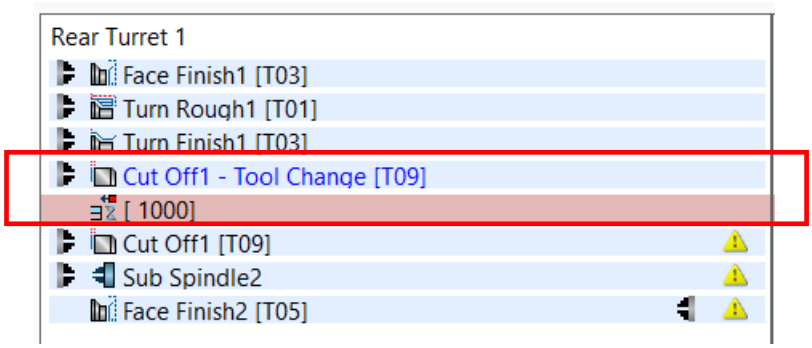
Wait Code when Start Location is 'Before the Start of the next operation'

Start Location Indicator for Wait Code when Wait Code Property is set to 'Before the First Rapid the next operation'

If the wait code property indicating the start position is set to ***Before the First Rapid of next operation***, then the wait code will be positioned before the tool change move of the operation before which the wait code has been inserted. An entry indicating the start location of the wait code will be displayed in the *Sync Manager* grid under the *Sync Manager* tab.

The syntax for the entry indicating the start position of the wait code will be:

<Name of the operation before which the wait code was inserted > - Tool Change [<Tool Station Number>]



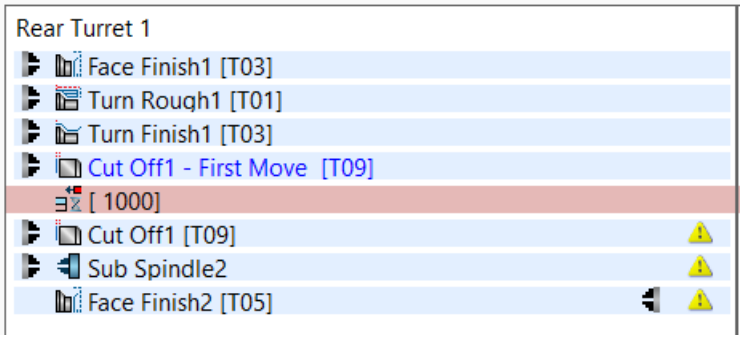
Start Location Indicator for Wait Code when Start Location is 'Before the First Rapid of next operation'

Start Location Indicator for Wait Code when Wait Code Property is set to 'Before the First Feed of next operation'

If the wait code property indicating the start position is set to ***Before the First Feed of next operation***', then the wait code will be positioned before the first non-cutting move of the operation before which the wait code has been inserted. An entry indicating the start location of the wait code will be displayed in the ***Sync Manager*** grid under the *Sync Manager* tab.

The syntax for the entry indicating the start position of the wait code will be:

<Name of the operation before which the wait code was inserted > - First Move [<Tool Station Number>]



Start Location Indicator for Wait Code when Start Location is 'Before the First Feed of next operation'

Notes on Start Location Indicators for Wait Codes

- They are for visual reference only and cannot be edited.
- They cannot be repositioned within the *Sync Manager* Grid.
- They are rendered in the color settings for suppressed operations.
- They do not appear in any *Setup Sheets*.
- They cannot be generated for post operations or Sub Spindle operations. (They are generated only for machining operations.)
- When executing the *Step Through* toolpath command, the step-through mechanism will consider all the start positions indicated for wait codes.
- When executing the *Simulate Toolpath* command, the simulation will consider all the start positions indicated for wait codes.
- When the toolpath is post processed, then the wait code is positioned as per its properties set in the Sync Manager.

Option to Create User Defined Folders under CAMWorks Trees

Purpose:

When working with complex parts or assemblies that contain multiple features, operations, or tools within the CAMWorks trees, navigating to the desired nodes can be challenging. Grouping features, operations, part setups, and tools with similar characteristics together can significantly enhance visual identification and management within the respective CAMWorks trees.

Implementation:

Starting with CAMWorks 2026, two new options, '**Create New Folder**' and '**Add to New Folder**,' have been introduced to enhance organizational capabilities within the CAMWorks Trees (i.e., Feature Tree, Operation Tree, and Tools Tree). These options allow you to create custom folders, enabling better grouping and management of Features, Operations, Part Setups, and Tools.

New Folder Options

The following options will be available as cascading menu items under the '**Folder**' option in the context menu for Feature nodes, Operation nodes, Part Setup nodes, and Tool nodes.

- **Create New Folder:** This option creates a new folder just above the selected node. Users can utilize the drag-and-drop functionality to add items to this newly created folder.
- **Add to New Folder:** This option creates a new folder just above the selected node and automatically includes the selected node within the folder. Additional items can be added using the drag-and-drop functionality.

User-defined Folder Options within the Feature Tree offer the following functionalities:

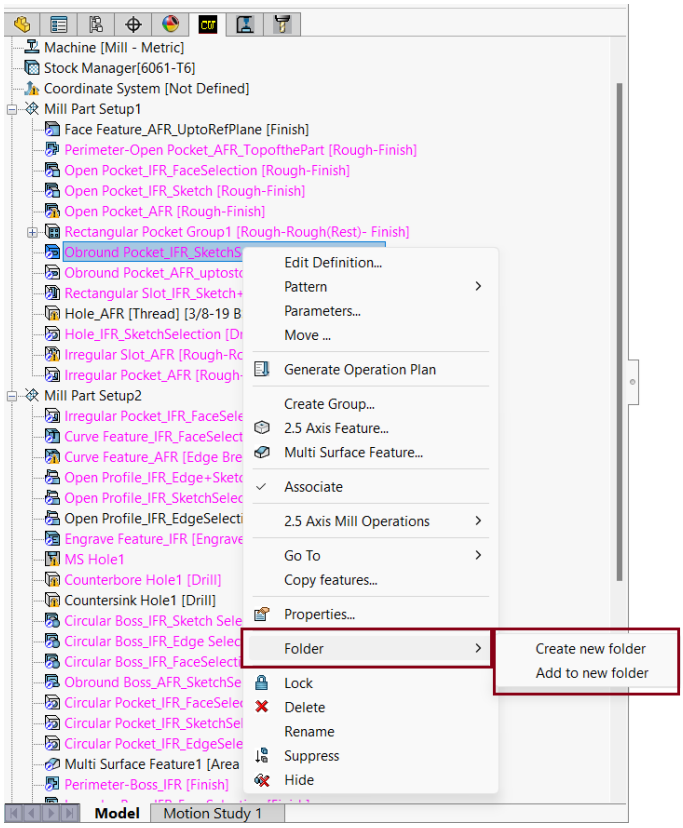
- Creation of folders at the Part Setup level
- Creation of folders at the Feature node level
- Grouping of Part Setups using drag-and-drop
- Grouping of Features with the same machining direction using drag-and-drop
- Relocation of user-defined folders within the Feature Tree using drag-and-drop

User-defined Folder Options within the Operation Tree offer the following functionalities:

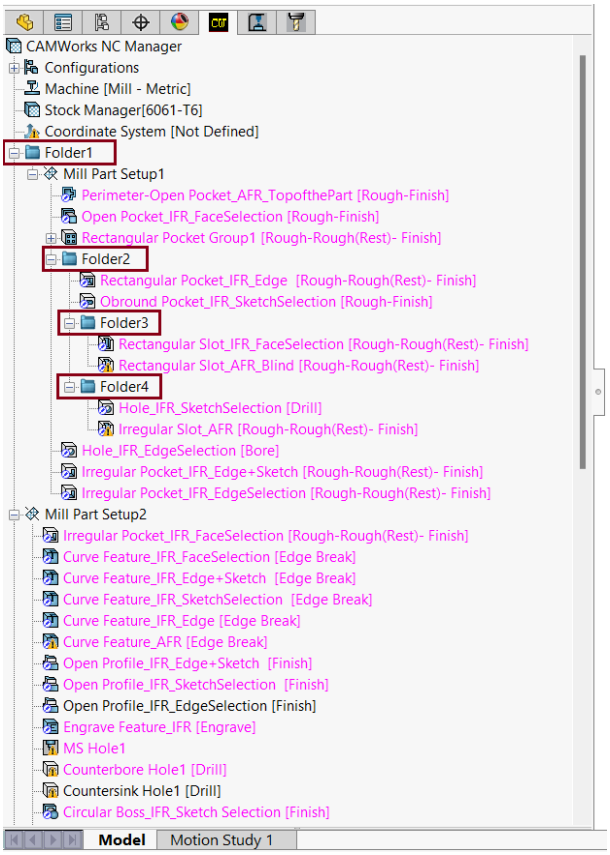
- Creation of folders at the Part Setup level
- Creation of folders at the Operation node level
- Grouping of Part Setups using drag-and-drop
- Grouping of Operations with the same machining direction using drag-and-drop
- Relocation of user-defined folders within the Operation Tree using drag-and-drop

User-defined Folder Options within the Tools Tree offer the following functionalities:

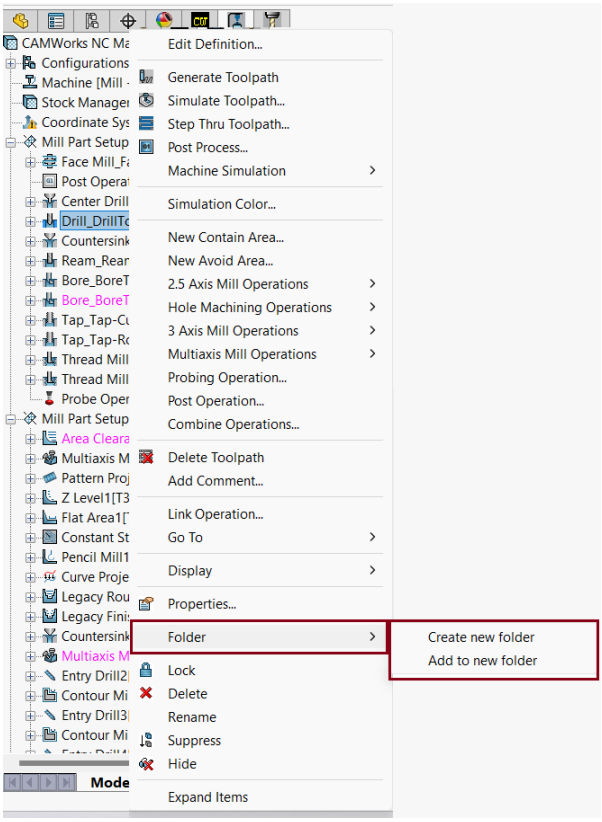
- Creation of folders at the Tool node level
- Grouping of Tool nodes using drag-and-drop
- Relocation of user-defined folders within the Tools Tree using drag-and-drop



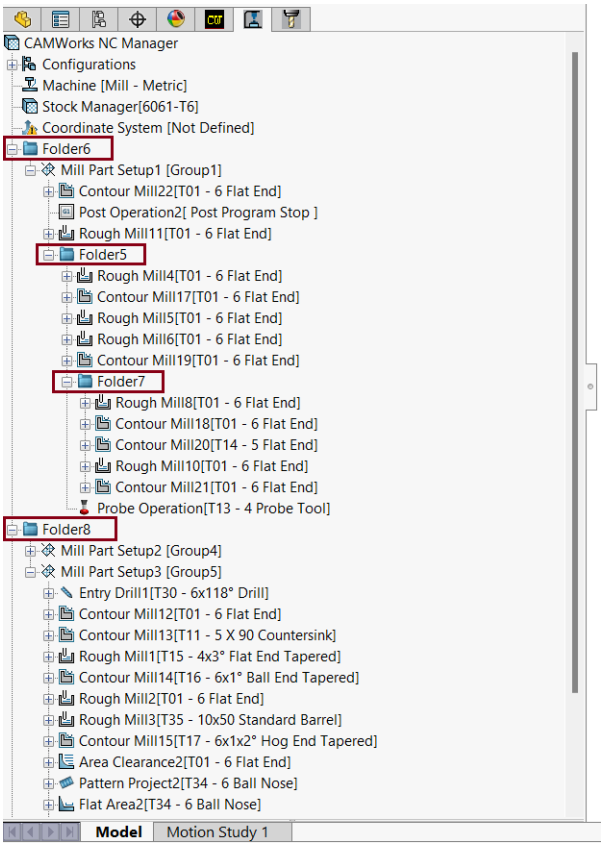
Context menu options 'Create New Folder' and 'Add To New Folder' under CAMWorks Feature tree



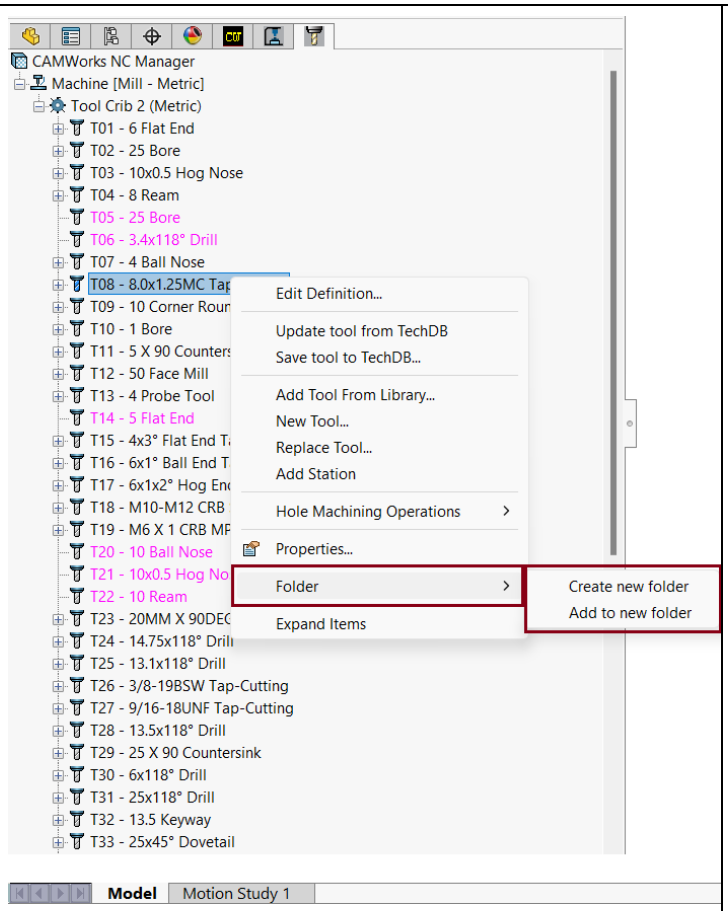
Example of User-defined folders created at multiple levels under the CAMWorks Feature tree



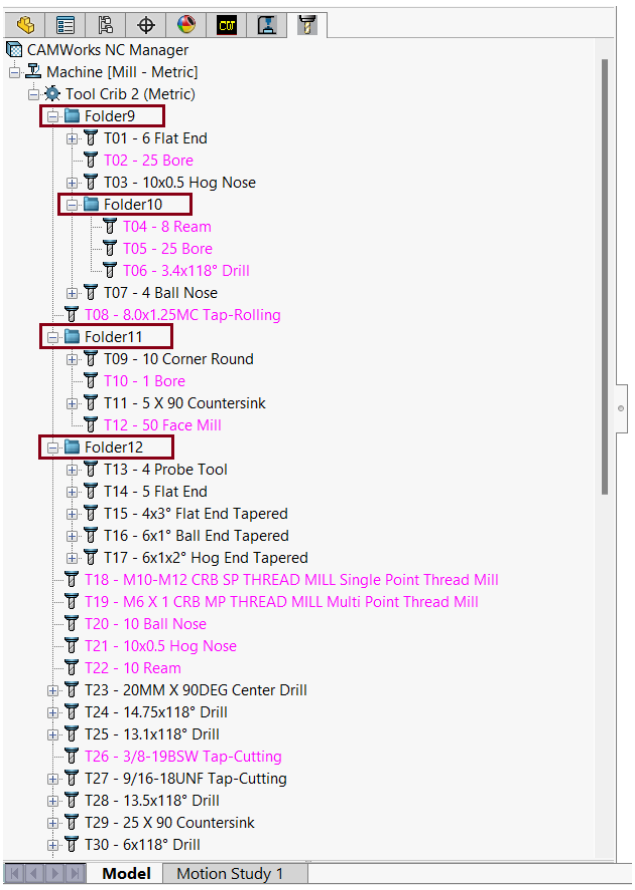
Context menu options 'Create New Folder' and 'Add To New Folder' under the CAMWorks Operation Tree



Example of User-defined folders created at multiple levels under the CAMWorks Operation tree



Context menu options 'Create New Folder' and 'Add To New Folder' under CAMWorks Tools Tree



Example of User-defined folders created at multiple levels under the CAMWorks Tools tree

Post Processors to generate NC code for Toolpaths Generated on Swiss Turn Machines

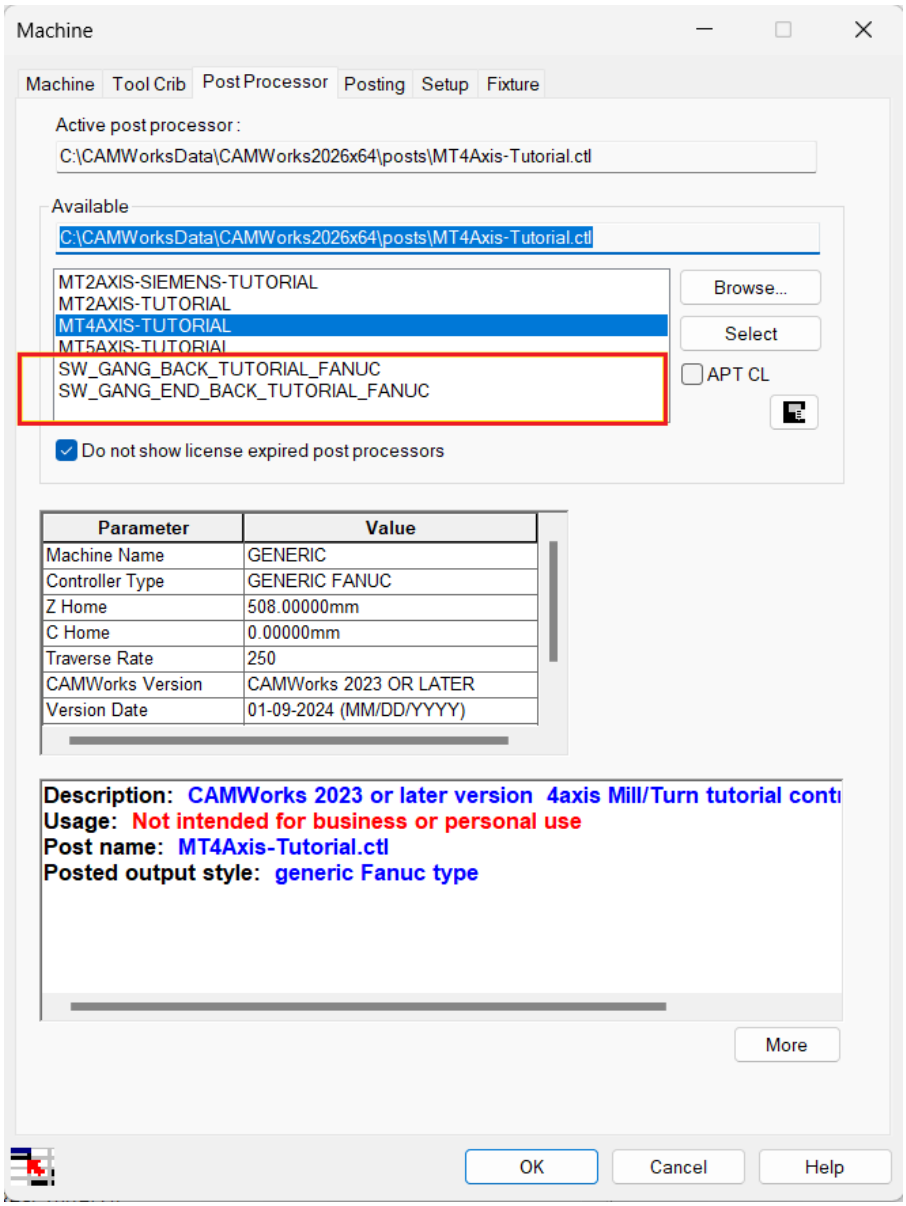
Purpose:

To ship post processors that support standard Swiss Turn machines in CAMWorks, thereby ensuring successful generation of G-code for toolpaths generated on Swiss Turn machines

Implementation:

For a part/assembly being machined, the post processor associated with the active machine under the *Post Processor* tab of the *Machine* dialog box plays a critical role in post processing the G-code generated for that part/assembly.

As multiple Swiss Turn and Multi-Function Lathe machine types are supported from *CAMWorks 2026* version onwards, the list of Post Processors shipped with CAMWorks has been expanded to include post processors that support generation of G-code for parts/assemblies on Swiss Turn machines.



New Post Processors that Support Post Processing of Toolpaths Generated on Swiss Turn/Multi-Function Lathe Machines

Custom Post Processors

In case you require custom post processors for your Swiss Turn/Multi-Function Lathe machines, we recommend that you contact your reseller for tailored customization options specific to your machining requirements.

Generation of G-Code for Parts programmed on Swiss-Turn Machines

Purpose:

Introduction of functionality to generate G-code in a new file format to enable viewing operations synchronized using wait codes

Implementation:

Most parts programmed on Swiss-Turn machines and Multi-Function Lathe machines usually contain wait codes. Adding wait codes allows the operations machined by the tools

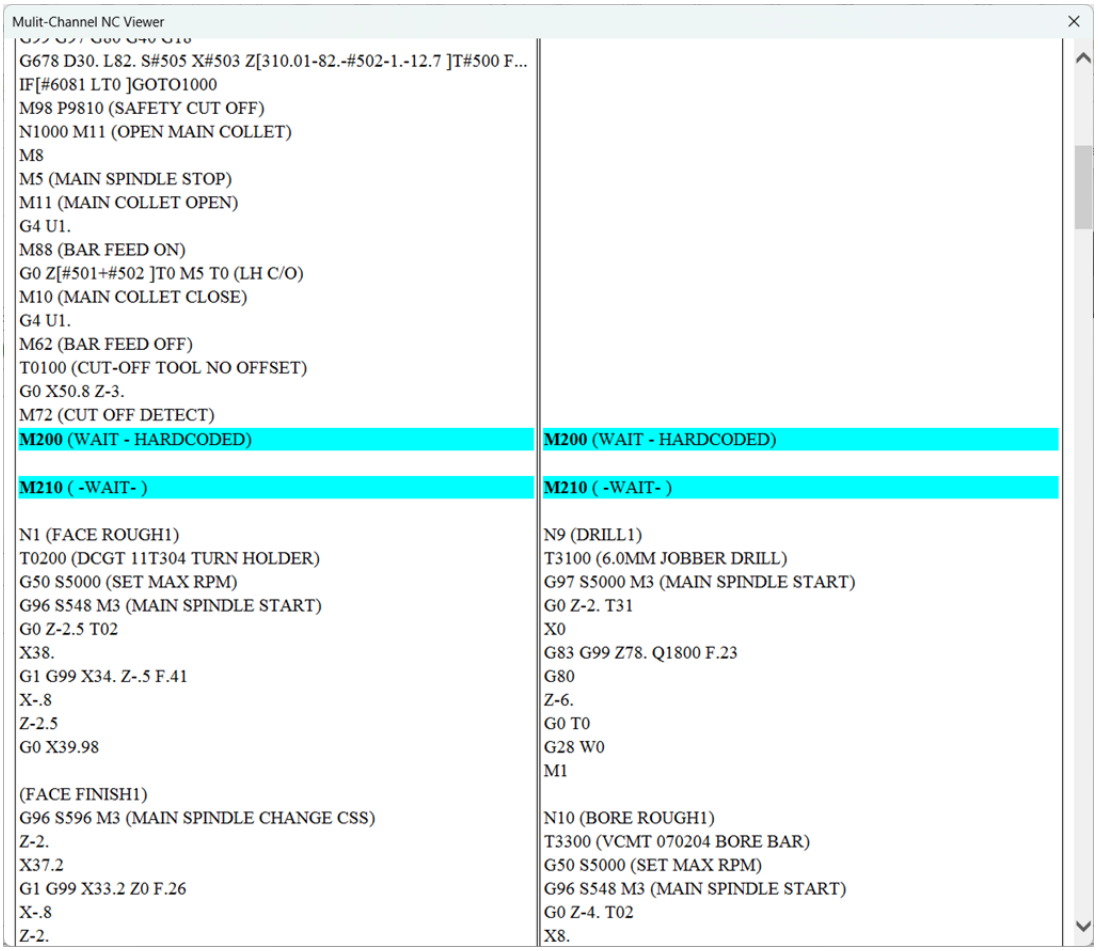
in the different tool posts to be synchronized. (This is achieved by re-ordering the operations in the *CAMWorks Sync Manager* user interface.) Optimal synchronization can drastically

reduce the time-required to machine the parts. The G-code generated for such parts will contain the wait codes before and/or after the operations they synchronize.

To ensure that a mechanism is available to view the operations synchronized by the wait codes within the generated G-code, an additional new file output format (viz. HTML) for generating G-code has been introduced. On executing the *Post process* command for a programmed part, the G-code will be output as both as a text file as well as an HTML file. Two HTML files will be generated viz. ***Aligned*** and ***Non-aligned***.

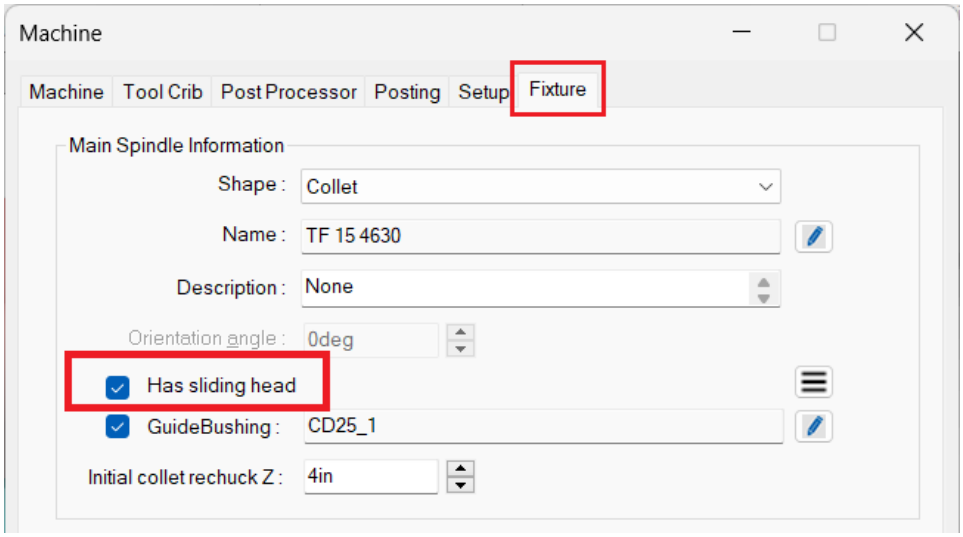
When the generated HTML file is opened, it will always be displayed in the ***Multi-Channel NC Viewer*** application. (This application will be auto installed when you install the *CAMWorks 2026* application.)

- In the HTML file of type *Aligned*, the wait codes will be synchronized. The NC code will be displayed in columns as per the number of tool posts in use for the operations. The wait codes will be highlighted within the columns to enable easy identification.
- In the HTML file of type *Non-aligned*, the G-codes will be displayed without the wait codes in different tool posts being aligned. The generated HTML file will have 'NonAligned' as suffix in its file name.



Sample G-Code HTML file of type 'Aligned' as viewed in Multi-Channel NC Viewer App

This functionality of generating G-code in HTML format will be available only for parts programmed for Swiss Turn Lathe or a Multi-Function Lathe machines. (The eligibility criterion being that the ***Has Sliding Head*** checkbox under *Fixture* tab of the *Machine* dialog box be checked for the CNC machine.)



'Has sliding head' Checkbox Option under Fixture Tab of the Machine Dialog Box

Note:

On executing the *Post process* command for parts programmed for Swiss Turn Lathe/Multi-Function Lathe machines, the G-code that will be output will continue to be output in default text file format (in addition to the two HTML files). The G-code file generated in default text file format can be opened, viewed and edited in the Editor application as indicated in the *'Open G-code file in*' field.

CAMWorks Tools Tree view under Tool Cribs Form of Technology Database

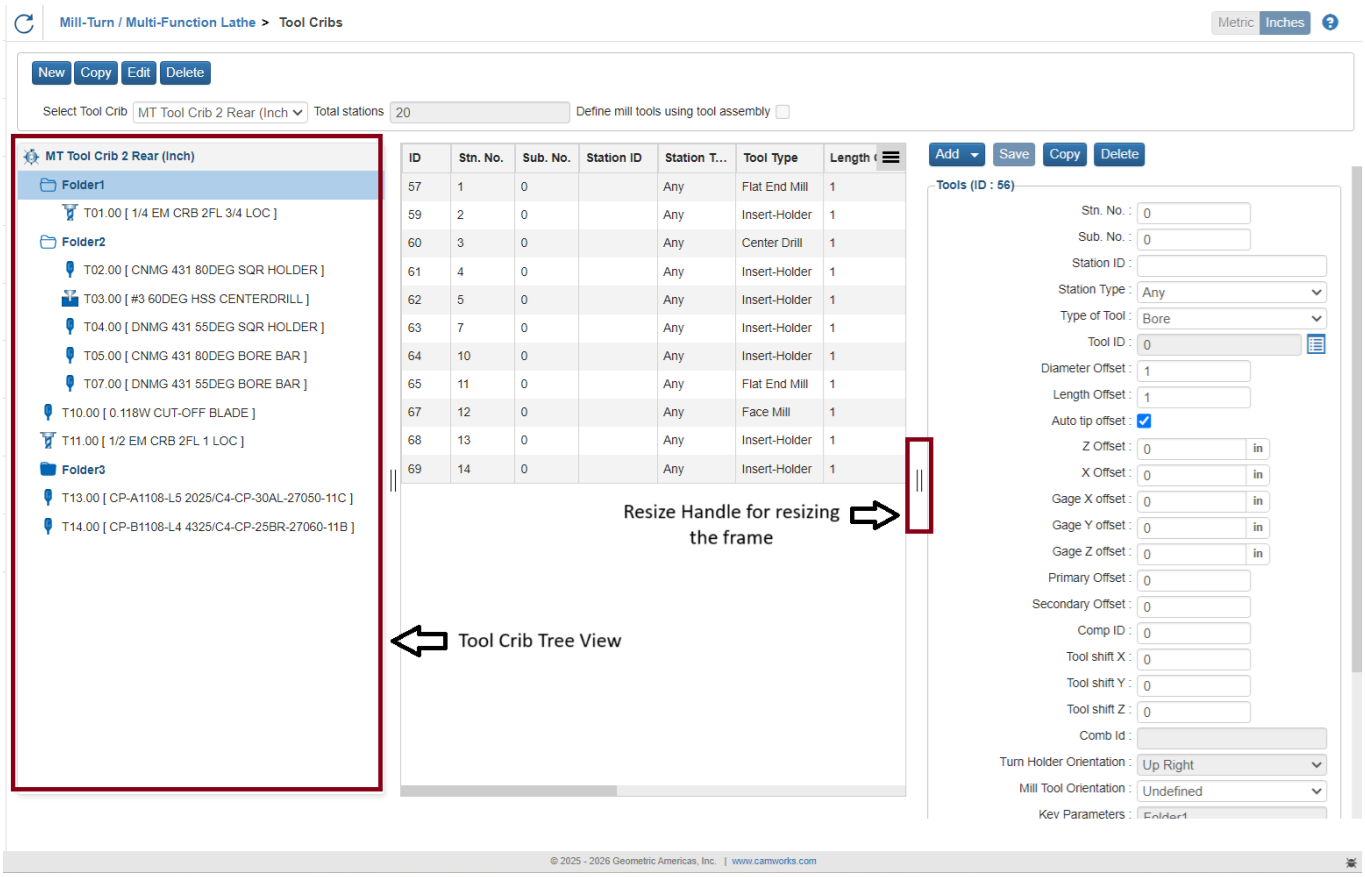
Purpose:

Enabling CAMWorks Tools Tree view under the Tool Cribs Form of Technology Database (TechDB)

Implementation:

A basic tree view has been added to the left-hand side of the grid under the ***Tool Cribs*** form of the CAMWorks Technology Database. This tree view will be identical to the one in the ***CAMWorks*** application, with the Tool Crib as the topmost node. This functionality mirrors the tree structure found in the CAMWorks application's interface in the Technology Database, thus fulfilling the aim of providing you with consistent and intuitive experience. The selection of tools has been synchronized between the tree view and the grid, meaning that selecting a tool in one will automatically select it in the other and vice versa.

The ***Tools Tree*** view introduced in the TechDB is a read-only field. However, you can resize the frames within this form, expand or collapse the folders with sub-items, retain the expand/collapse state of folders on refresh, and expand the tree if the selected tool is inside a minimized folder. The Tree View will be displayed for all three modules: Mill, Turn, and Mill-Turn/Multi-Function Lathe.



Tools Tree View Under the Tool Cribs Form of CAMWorks Technology Database (TechDB)