

# What's New in



# CAMWorks

2024



HCL Technologies Ltd. makes no warranties, either expressed or implied with respect to this document. HCL reserves the right to revise and improve products as it sees fit, and to revise the specifications and information contained herein without prior notice. Due to continuing product development, specifications and capabilities described in this document are subject to change without notice.

**Copyright © 2023 HCL Technologies Ltd. All Rights Reserved.**

**September 2023**



# Table of Contents

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

<b>What’s New in CAMWorks 2024 BETA</b>	<b>3</b>
Supported Platforms	3
Resolved CPRs Document	3
<b>General</b> .....	<b>4</b>
Dialog Box showcasing Legend during Toolpath Simulation	4
<b>TechDB</b> .....	<b>5</b>
Option to assign default Virtual Simulation Machine to be associated with a saved Machine in TechDB	5
<b>Mill</b> .....	<b>7</b>
Toolpath Output Options for Bore, Dovetail, Keyway, and User- Defined Tools	7
Option to Suppress Fillets and Chamfers of Mill Features for VoluMill Operations	9
Behavior of ‘Link to Tool’ checkbox option under F/S tab of Operation Parameters Dialog Box	11
‘Avoid Part Faces; Option 2.5 Axis VoluMill Toolpaths	13
Option to define Cut Amounts as Percentage of Flute Length for 2.5 Axis Mill Operations	15
Optimized Pecking Option for Drill Toolpaths generated for Condensed Hole Features	17
Improved Options for Condensed Hole Features	19
Dynamic Highlighting of Nodes in Feature Tree and Operation Tree	22
Improved Leadin/Leadout for Thread Milling Toolpaths for Hole Features	25
<b>Turn / Mill-Turn</b> .....	<b>26</b>
Options to Highlight Recognized/Unrecognized Faces of the Turn Part Models	26
Defining SOLIDWORKS Fixture Coordinate System (FCS) in Turn Assembly Mode	27
Option to Create Centralized Library of Turn Tool Blocks in TechDB	28
Option to start programming of Turn/Mill-Turn parts by holding them in Sub Spindle	31
<b>Multiaxis</b> .....	<b>32</b>
Tilt Tool by Contact Point option for Multiaxis Mill Operations	32
Multiaxis Machining – Roughing Toolpath for Roughing out Pocket Geometries	34
Multiaxis Machining - Finishing Toolpaths for Finish Machining Pocket Geometries	36
<b>Licensing</b> .....	<b>39</b>
Message Prompt to upgrade License Manager on Floating Networks During CAMWorks Upgrade	39



# What's New in CAMWorks 2024 BETA

## Supported Platforms

Supported Platforms for 64-bit	
<b>Solid Modeler:</b>	<p>The 64-bit version of:</p> <ul style="list-style-type: none"> <li>- SOLIDWORKS 2023</li> <li>- CAMWorks Solids 2023</li> </ul> <p><b>Note:</b> SOLIDWORKS 2024 and CAMWorks Solids 2024 will be supported once released.</p>
<b>Operating System:</b>	<p>64-bit version of:</p> <ul style="list-style-type: none"> <li>- Windows 11</li> <li>- Windows 10</li> </ul> <p style="text-align: right;">[*Home Editions are not supported]</p> <p><b>Note:</b> CAMWorks 2024 is supported only on 64-bit Operating systems.</p>

## Resolved CPRs Document

<b>Purpose of Document:</b>	The Resolved CPR ( <i>CAMWorks Problem Report</i> ) document has been updated to report the software errors that have been resolved in the current Service Pack (BETA).
<b>Path to Document:</b>	<p>To view the document, select:</p> <p><i>C:\Program Files\CAMWorks2024x64\CAMWorks_VC142\Lang\English\CW2024.pdf</i></p>



# General

## New - Dialog Box showcasing Legend during Toolpath Simulation

**Purpose:**

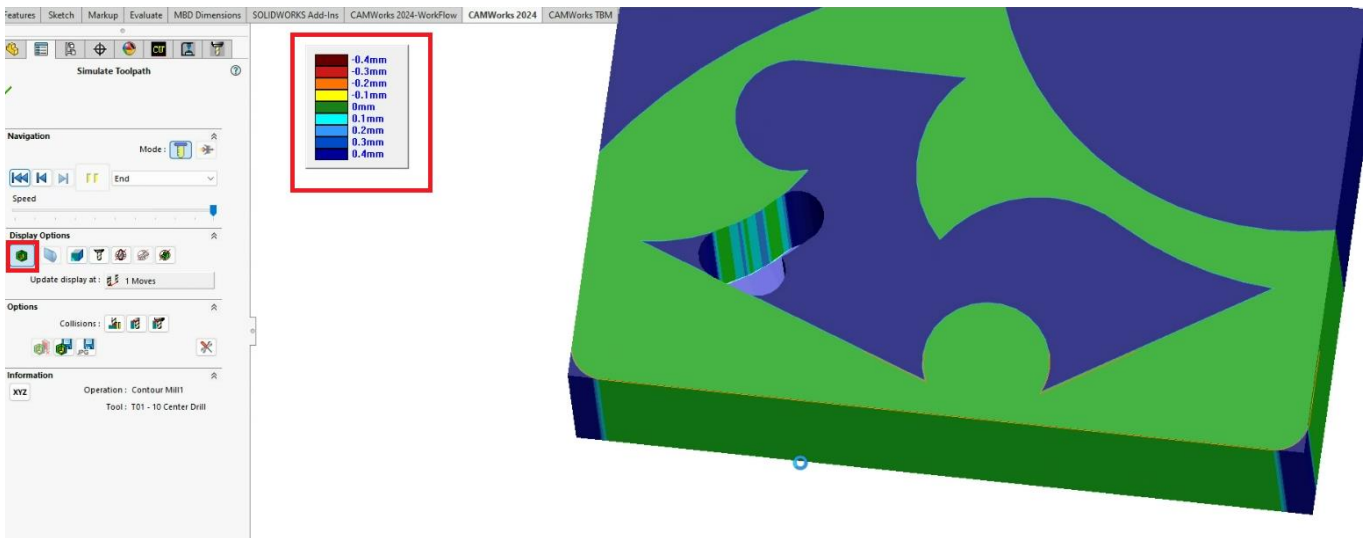
To provide a dockable dialog box indicating the legend for graphical comparison of the machined part and the design part during toolpath simulation

**Implementation:**

When the Show difference command on the Simulation toolbar was executed, a graphical comparison of the machined part and the design part is displayed. The amount of deviation is identified by different colors on the simulated model.

In previous versions of *CAMWorks*, a legend at the top right corner of the graphics area listed these various colors and the corresponding deviation indicated by those colors. The legend was always displayed in the upper left corner and couldn't be relocated.

From CAMWorks 2024 version onwards, this legend will be displayed in a dockable dialog box that can be moved to any other desired location within the graphics area.



Legend indicating deviation between machine dpart and design part displayed in a dockable window



# TechDB

## **New - Option to assign default Virtual Simulation Machine to be associated with a saved Machine in TechDB**

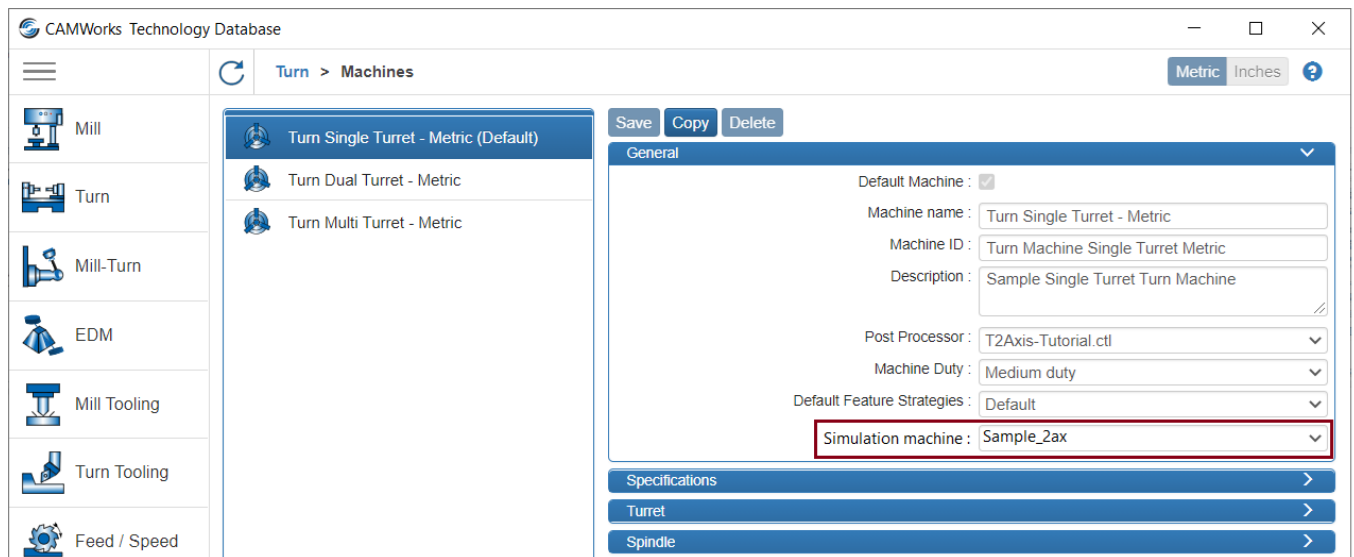
### **Purpose:**

To provide the option to assign the default virtual simulation machine that will be associated with the selected Mill, turn or Mill-Turn machine in the TechDB

### **Implementation:**

The **Simulation machine** dropdown list has been introduced in the **General** tab of the Mill, Turn and Mill-Turn Machine User Interfaces within the TechDB.

This dropdown field is used to assign the default virtual simulation machine that will be associated with the selected machine. To assign another simulation machine, click on the **Simulation machine** field. A dropdown list of all the simulation machines compatible with the currently selected machine will be listed. Select the simulation machine from this list and click on the **Save** button.



### **Default Simulation Machine in 'Simulation Machine' Dropdown list**

Under default TechDB settings, the Simulation machine dropdown list will be populated with the virtual simulation machines shipped with CAMWorks. Once CAMWorks is installed, the default folder path to these virtual simulation machines will typically be:

**C:\CAMWorksData\CAMWorks202Xx64\MachSim\Machines**

Under default TechDB settings, for a selected machine in the Machines user interface, the **Simulation** machine dropdown list will populate virtual simulation machines from the above folder location (and its sub-folders) provided that those simulation machines are compatible with the selected machine.

### **Populating 'Simulation Machine' Dropdown list with Customized Simulation Machines**

If the default simulation machines shipped with CAMWorks do not suit your machining requirements, you can replace them with customized virtual simulation machines as per your requirements.

In order to assign a customized virtual simulation machine to a machine selected in the TechDB, it is necessary that the **Simulation machine** dropdown list be populated with the customized simulation machines instead of the default simulation machines. To populate it with customized simulation



machines, the folder location from where the simulation machines are picked needs to be changed. Use the ***Simulation machine path*** field under the ***General*** tab of ***Settings*** user interface in TechDB to specify the folder path to the customized simulation machines. Once this *Simulation machine path* field is updated, the ***Simulation machine*** dropdown list will be populated with customized simulation machines present within that folder and its sub-folders, provided that they are compatible with the currently selected machine.

### Assigning a Simulation Machine for a Machine within the CAMWorks User Interface

In the CAMWorks user interface, for machining part models and assemblies, machines defined in the TechDB will be available for selection in the ***Machine*** tab of the ***Machine*** dialog box. For a selected machine, the default simulation machine assigned will be the selection made in the Simulation machine dropdown list of TechDB.

However, if required, you can reassign a different simulation machine to the selected machine using the ***Simulation machine*** dropdown list available under the Machine tab of the Machine dialog box within the CAMWorks application.



# Mill

## Improved - Toolpath Output Options for Bore, Dovetail, Keyway, and User- Defined Tools

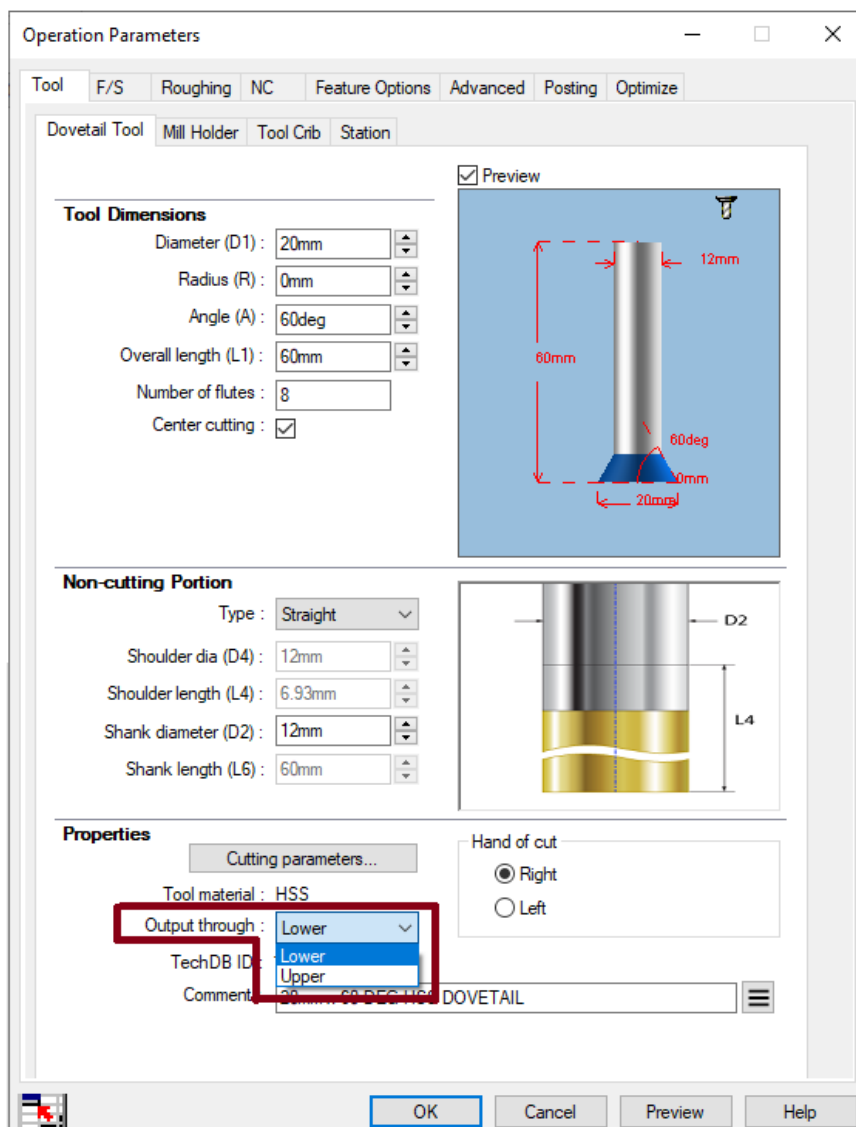
### Purpose:

For 2 Axis and 3 Axis toolpaths that use Dovetail, Keyway, Bore and User-defined Tools, to provide the additional option of outputting the toolpath using the flute length of tool in addition to the default setting of outputting the toolpath through the tool tip.

### Implementation:

In previous versions of CAMWorks, for all 2 Axis Mill and 3 Axis Mill operations that supported Keyway tools, Bore Tools, Dovetail Tools or User-Defined Mill Tools, the toolpaths generated could only be output through the tool tip. No option to alter this output was available.

From CAMWorks 2024 BETA version onwards, users will be provided with the options to choose how the toolpath will be output through. A new dropdown list named **Output Through** has been added under the **Tool** page under the corresponding *Operation Parameters* dialog box for operations perusing the aforementioned tools.



**New 'Output Through' Option present under Tool Tab for Operations that Use Bore, Dovetail, Keyway or User-Defined Tools**



**Options in the 'Output Through' Dropdown List**

Use this dropdown list option to specify the reference point for outputting G-Code coordinates for that specific operation. Following are the two options available within this dropdown list:

- **Lower:** This is the default option. When selected, the G-code for the toolpath of the specific operation will be output through the tool tip.
- **Upper:** When this option is selected, the Z coordinates for the toolpath will be compensated using the flute length of the tool.

<p><b>Illustrative Example of Dovetail Tool when Output Through Option is set to 'Lower'</b></p>	<p><b>Illustrative Example of Dovetail Tool when Output Through Option is set to 'Upper'</b></p>

**Note:**

If a Keyway tool, Bore Tool, Dovetail Tool or User-Defined Mill Tool is assigned to a *Face Mill* operation, then this **Output Through** option under the corresponding Tool page will be set to **Lower** and disabled. It will remain disabled for all *Face Mill* operations using any one of the previously mentioned tools as *Face Milling* only supports toolpath output through the tool tip.





**New - Option to Suppress Fillets and Chamfers of Mill Features for VoluMill Operations**

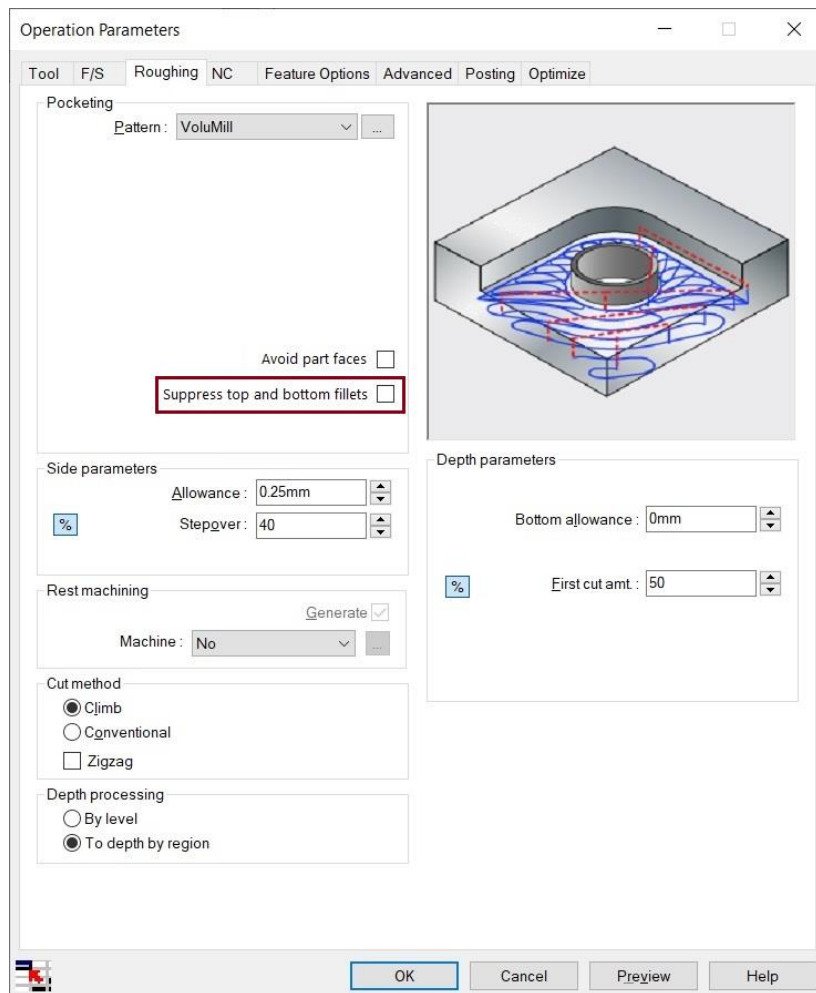
**Purpose:**

To provide an option to suppress the chamfers and fillets of a 2.5 Axis Mill Feature being machined using a VoluMill operation so that only vertical faces are considered when generating the VoluMill toolpath

**Implementation:**

In previous versions of CAMWorks, if VoluMill toolpaths were generated for 2.5 Axis Mill features with chamfers and/or fillets, then the toolpath generated considered the geometry of the chamfers and/or fillets. An option to ensure that chamfers/fillets were ignored and only the vertical faces were considered while computing the VoluMill toolpath was not available. (Ignoring the chamfers and fillets is necessary if a user wishes to assign a different operation and machining conditions for machining these chamfers and fillets.)

From CAMWorks 2024 BETA version onwards, to address this issue, a checkbox option labeled **Suppress top and bottom fillets** will be available under the *Roughing* tab of *Rough Mill* operations.



**New 'Suppress top and bottom fillets' checkbox option under Roughing Tab for Rough Mill Operations**

This parameter is displayed under the *Roughing* tab only when the *Pocketing Pattern* is set to **VoluMill**. Under default settings, this checkbox remains unchecked. These default settings can be changed in TechDB within the *Rough Mill Form* available under *Default Operation Parameters* option available in the *Mill* menu.



## What's New in CAMWorks 2024 BETA

The screenshot shows the CAMWorks 2022 Technology Database interface. The left sidebar contains navigation options: Mill, Turn, Mill-Turn, EDM, Mill Tooling, Turn Tooling, Feed / Speed, Settings, and About. The main window displays the 'Mill > Default Operation Parameters' for a 'Rough Mill' feature. A table lists parameters with columns for Id, Name, and Description. The 'Roughing' tab is active, showing various settings for 'Pocketing' such as Pattern (VoluMill), Cut angle (0 deg), Start Corner (Top Right), Minimum Spiral Diameter (5 mm), Cycle Type (Pecking), First peck amt (5 mm), Sub. peck amt (5 mm), Step distance (50 %), Min. distance (10 %), Cleanup pass (checked), Stay at depth (unchecked), and Machine island top (checked). The 'Suppress top and bottom fillets' checkbox is highlighted with a red box and is currently unchecked.

Id	Name	Description
79	Default	Default - Metric
135	Cavity machining	Pocket out machining
136	Core machining	Pocket in machining
137	Volumill	Volumill machining

### Suppress top and bottom fillets checkbox option under Roughing Tab in Rough Mill Form of TechDB

When this checkbox option is checked, the Z machining depths will be made as if the sides of the feature extended to the top and/or bottom of the feature.

**Note:**

This option is applicable only for fillets and chamfers for the side walls of a 2.5 Axis Mill feature. It is not applicable for chamfers and fillets on islands present within the mill feature.



**Improved - Behavior of 'Link to Tool' checkbox option under F/S tab of Operation Parameters Dialog Box**

**Purpose:**

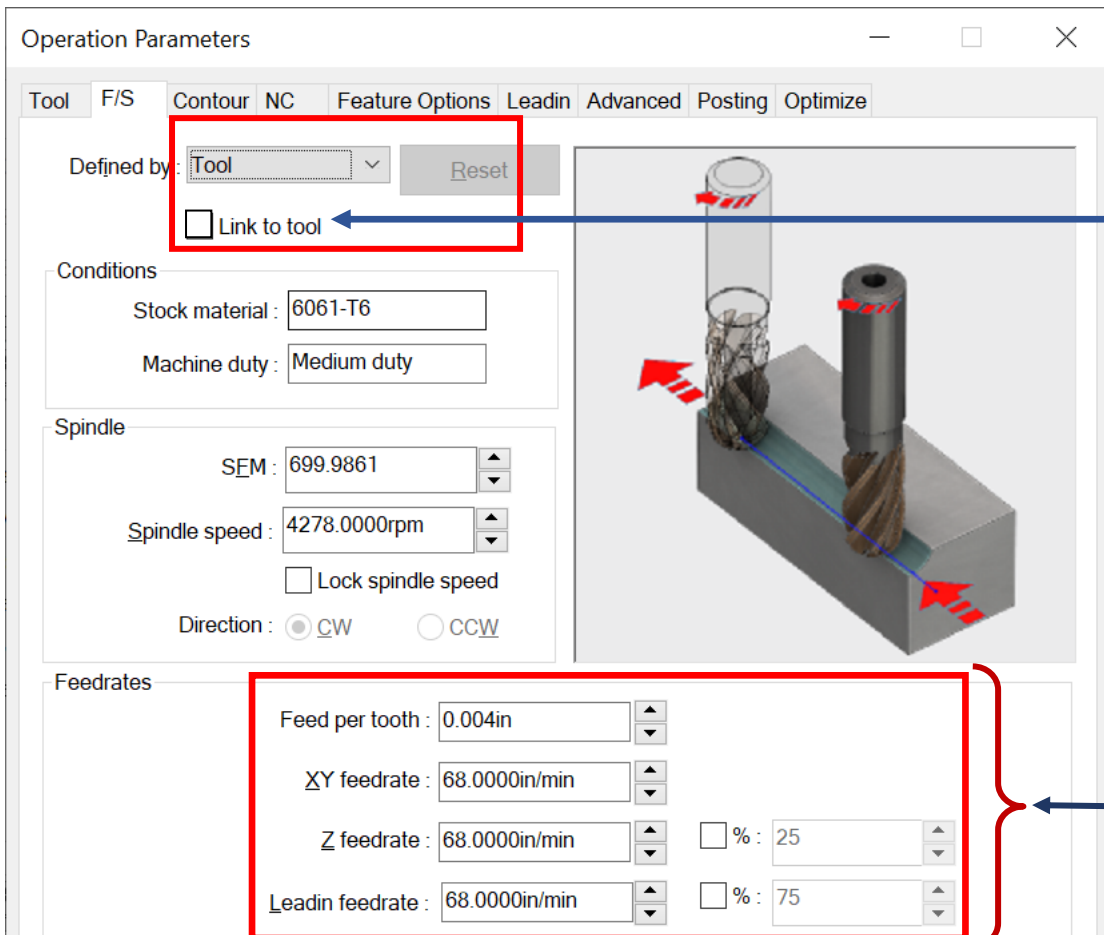
For 2 Axis and 3 Axis toolpaths that use Dovetail, Keyway, Bore and User-defined Tools, to provide the additional option of outputting the toolpath using the flute length of tool in addition to the default setting of outputting the toolpath through the tool tip.

**Implementation:**

The **Link to tool** checkbox is displayed under the **F/S** tab only when **Tool** is selected in the **Defined by** dropdown list. When this option is checked, the feed/speed parameters under this tab are feed/speed values associated with the assigned to tool's cutting parameters. Users cannot override these values as the feed /speed parameter fields get disabled when this checkbox is checked.

In previous versions of *CAMWorks*, there was no option to uncheck the **Link to tool** checkbox within the *CAMWorks* user interface. The checkbox would be checked even if the default setting for it in the corresponding **Operation Parameters** form within the TechDB was assigned the unchecked status. The option to uncheck this checkbox and override/edit the feed and speed values associated with cutting parameters of the tool was not available.

From **CAMWorks 2024 BETA** version onwards, the behavior of the **Link to tool** checkbox has been altered to eliminate this limitation. Users will now be able to check/uncheck this checkbox within the **CAMWorks** user interface. The checked or unchecked status of this checkbox will now be linked to its default setting in the TechDB. Additionally, users can check/uncheck this checkbox within the *CAMWorks* user interface.



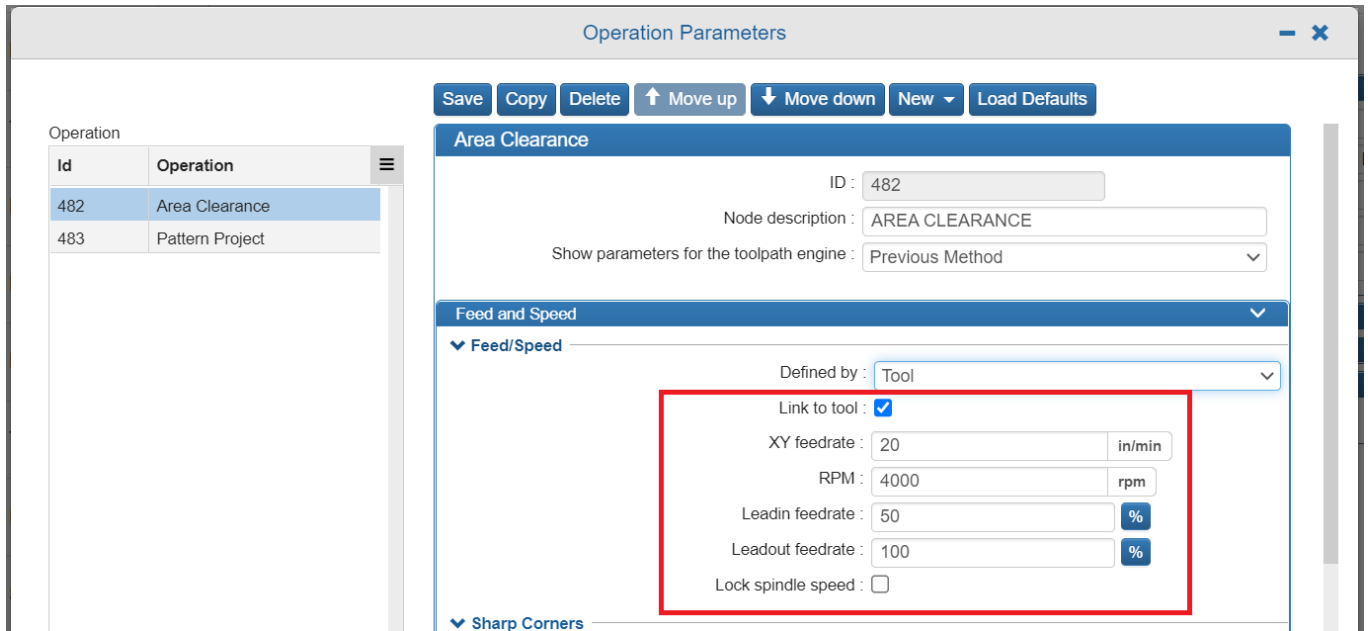
'Link to Tool' checkbox can now be checked or unchecked within the user interface.

When 'Link to Tool' is unchecked, feed/speed parameters will be assigned default values as assigned in TechDB. These values can be edited. For legacy parts, the feed/speed values previously saved with the part will be displayed.

**F/S Tab under Operation Parameters Dialog Box for Mill Operations**



In the **Operation Parameters** form in the TechDB, when this option is unchecked under its **Feed and Speed** tab, the feed/speed parameters under that tab will be enabled. Assign the desired values to these parameters and save the changes. These user-assigned default values will then be correspondingly reflected under the **F/S** tab within the **CAMWorks** user interface where they can be edited and saved as per user requirements.



**Feed and Speed Tab in Operations Parameters Form within TechDB User Interface**

**Note:**

For legacy parts/assemblies (viz. parts/ assemblies programmed using a CAMWorks version prior to **CAMWorks 2024 BETA**), the behavior of this functionality will be slightly different. When the **Link to tool** option is unchecked within the **CAMWorks** user interface, there will be no corresponding feed/speed values saved within the TechDB to populate the now enabled feed/speed parameters under the F/S tab. These parameters will be assigned the feed/speed values originally saved with the legacy par when it was programmed.

Users can change these values and save these updated values as per their requirements.



**New - 'Avoid Part Faces; Option 2.5 Axis VoluMill Toolpaths**

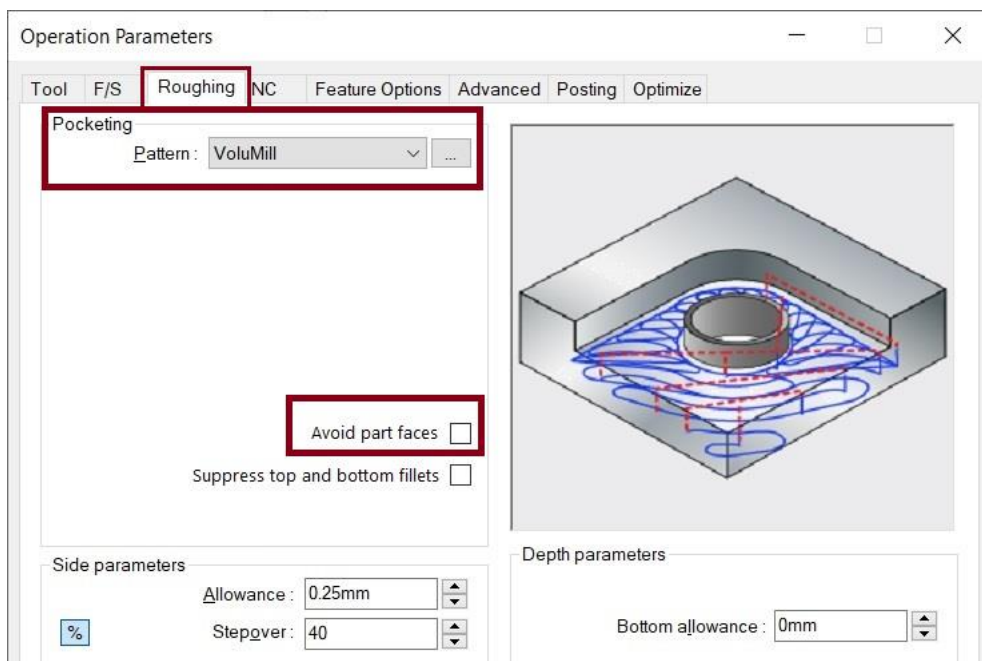
**Purpose:**

If faces of other mill features overlap with the mill feature being machined with a VoluMill toolpath, then to provide an option that ensure that such overlapping faces are treated as faced to be avoid by the VoluMill toolpath.

**Implementation:**

In previous versions of CAMWorks, the VoluMill toolpath generated for 2.5 Axis Mill features did not consider any overlapping faces of other mill features on the part model as faces to be avoided. Consequently, the VoluMill toolpath occasionally ended up gouging the part being machined.

From CAMWorks 2024 BETA version onwards, a new checkbox option named **Avoid part faces** has been introduced under the **Roughing tab** for **2.5 Axis Rough Mill** operations. This checkbox will be visible in the user interface only when **VoluMill** is selected as the **Pocketing Pattern** under the same tab.



**New 'Avoid part faces' checkbox option under Roughing Tab for Rough Mill Operations**

- When this checkbox option is unchecked, the VoluMill toolpath generated for the mill feature(s) will not consider any overlapping faces of other mill features on the part model as faces to be avoided. Consequently, the VoluMill toolpath might end up gouging the part.
- When this checkbox option is checked, all other faces of the part model (except the faces of the mill feature(s) being machined by the VoluMill toolpath) will be considered as avoid faces. The VoluMill toolpath generated will avoid those faces.

**How 'Avoid part faces' Option works for VoluMill toolpath generated for Combined Features**

If the VoluMill toolpath is generated for combined Mill features, then when machining one of the features in the group, the toolpath will treat the faces of other mill features in the same group as avoid features as well as the other faces of the part model that do not belong to the combined Mill features group.

**How 'Avoid part faces' Option works for VoluMill toolpath generated for an Assembly**

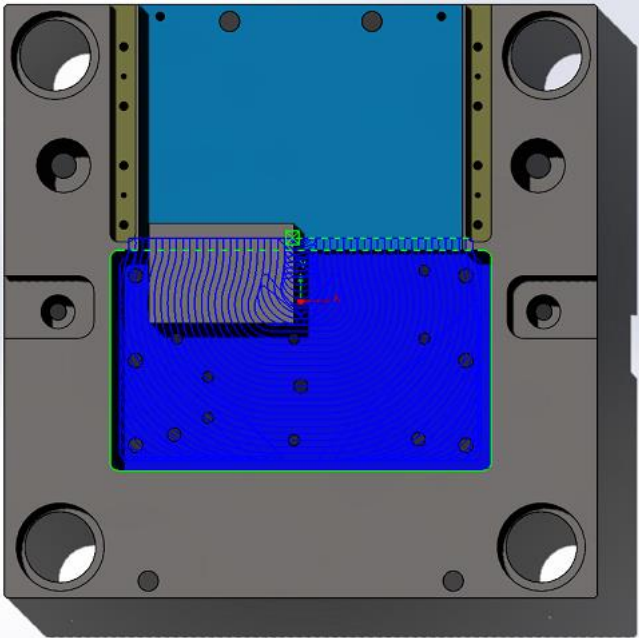
If the VoluMill toolpath is generated for Mill features on any component part of an Assembly, then only eligible faces of that component part will be considered as avoid faces by the VoluMill toolpath when the **Avoid part faces** checkbox is checked.

The faces of other parts of the assembly will not be considered as Avoid features.

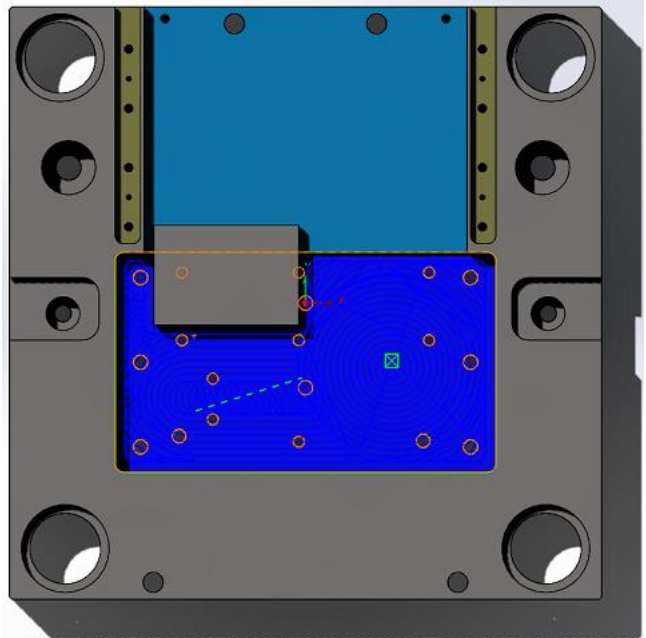


**How 'Avoid part faces' Option works when one or more Avoided Faces are Suppressed**

When the **Avoid part faces** checkbox is checked and the VoluMill toolpath is generated for the Mill feature(s) of any part model/assembly, only faces of the part/assembly that are not suppressed will be considered as faces to be avoided. If any face that was avoided by the VoluMill toolpath is suppressed after the generation of the VoluMill toolpath, then CAMWorks will display a **Rebuild** pop-up message prompting you to rebuild the part/assembly. When this checkbox option is checked, the Z machining depths will be made as if the sides of the feature extended to the top and/or bottom of the feature.



**VoluMill toolpath when 'Avoid Part Faces' option is unchecked for pocket feature. (Observe that face of the overlapping Rectangular Boss feature is not avoided VoluMill toolpath.)**



**VoluMill toolpath when 'Avoid Part Faces' option is checked for the pocket feature. (Observe that face of the overlapping Rectangular Boss feature is now avoided by the VoluMill toolpath.)**



**New - Option to define Cut Amounts as Percentage of Flute Length for 2.5 Axis Mill Operations**

**Purpose:**

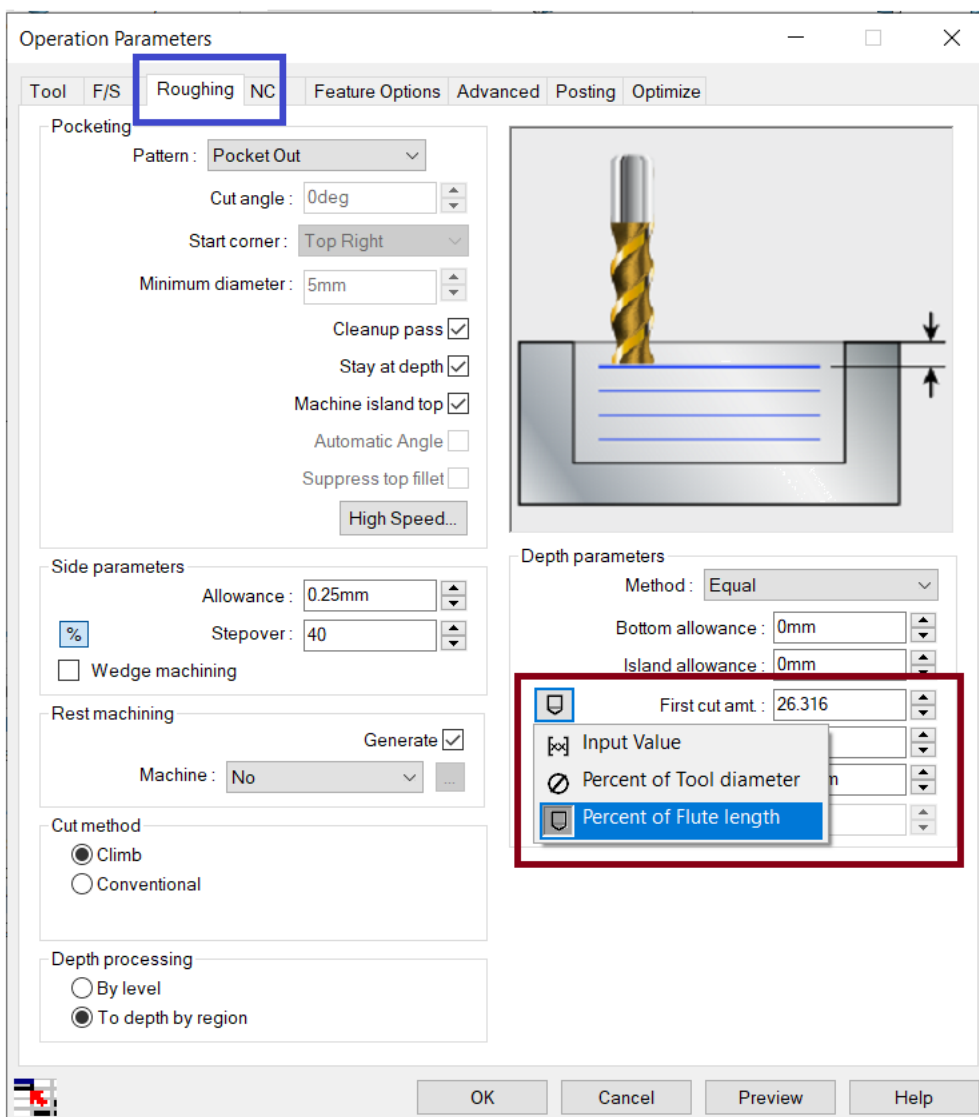
To provide an option that allows users to define *Cut Amount* parameters associated with 2.5 Axis Mill operations to be defined as a percentage of their assigned tool's flute length

**Implementation:**

In previous versions of *CAMWorks*, for Rough Mill, Contour Mill and Face Mill operations, **Cut Amount** parameters such as **First Cut Amount**, **Max Cut Amount** and **Sub Cut Amount** could be defined as absolute values or as a percentage of the tool diameter.

From *CAMWorks 2024 BETA* version onwards, an additional option to define these *Cut Amount* parameters as a percentage of the **Flute Length** will be available. This option will be available for **Cut Amount** parameters in the following user interfaces:

- **First Cut Amount**, **Sub Cut Amount** and **Max Cut Amount** parameters within *Depth Parameters* group box under Rough Mill tab for **Rough Mill** operations.
- **First Cut Amount**, **Sub Cut Amount** and **Max Cut Amount** parameters within *Depth Parameters* group box under **Contour** tab for **Contour Mill** operations.
- **First Cut Amount** and **Max Cut Amount** parameters within *Depth Parameters* group box under **Facing** tab for **Face Mill** operations.






**Option to Define Cut Amount Parameters as 'Percent of Flute Length' under Contour Tab UI**






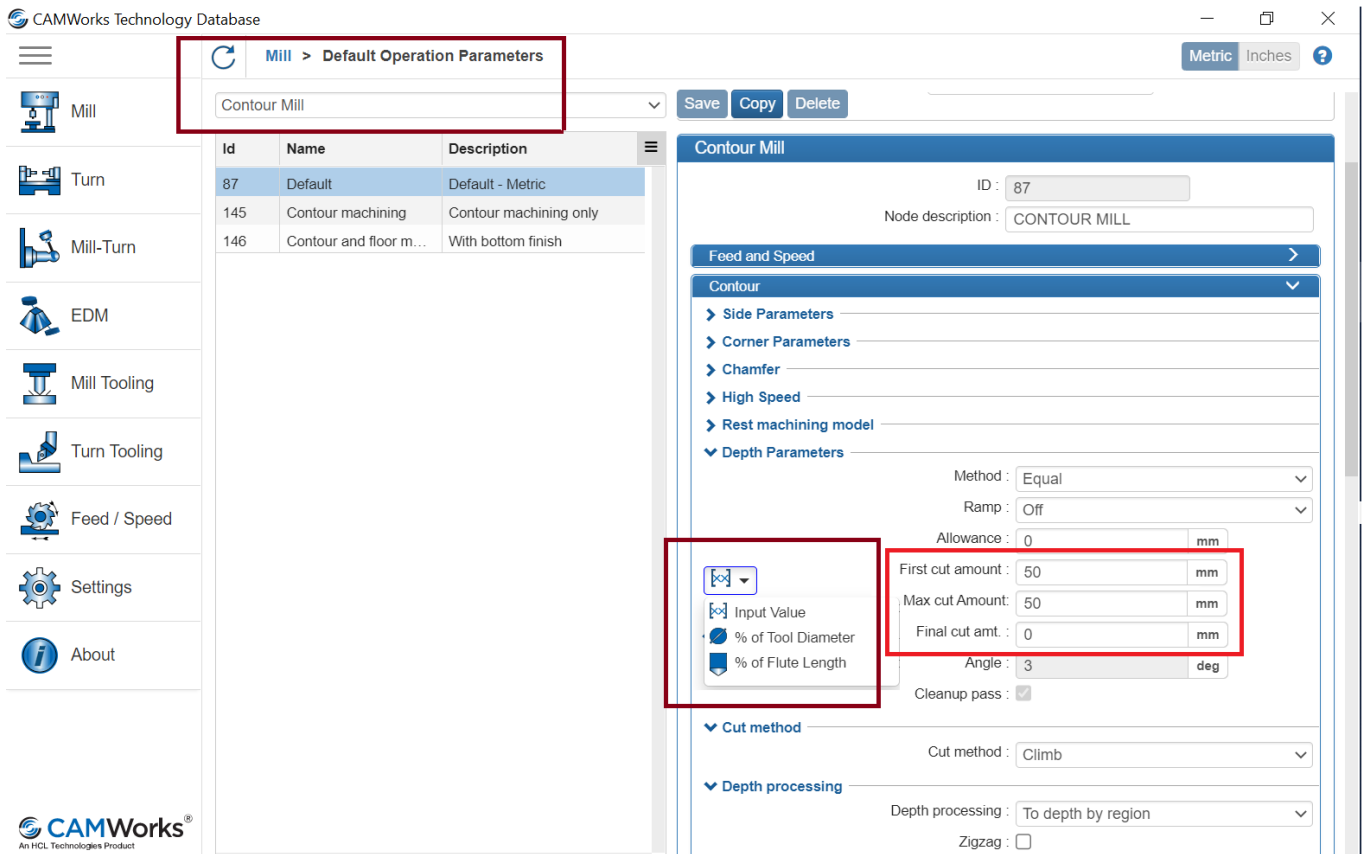
Within the above-listed user interfaces, the previously present **Percentage** toggle button to the left of the *Cut Amount* parametric fields within *Depth parameters* group box has been replaced with a flyout button. Clicking on this button displays three options. Select the desired option for determining the method for assigning the **Cut amount** value.

Following are the available options when this flyout button is clicked:

- **Input Value** : When this option is selected, the cut amount parametric value will be defined as an absolute value. This is the default option.
- **Percent of Tool Diameter** : When this option is selected, the cut amount parametric value will be defined as a percentage of the assigned tool's diameter.
- **Percent of Flute Length** : When this option is selected, the cut amount parametric value will be defined as a percentage of the assigned tool's Flute Length.

**Implementation within the TechDB User Interface**

The default values for **Cut Amount** parameters can be saved within the corresponding *Operation Parameters* form within the Technology Database App. In the user interface within the TechDB wherever the *Cut Amount* parameters can be edited, the flyout buttons that display options to save the *Cut Amount* values as absolute values , percent of tool diameter  and percent of flute length  will be displayed. Use these options to edit/save the desired cut amount default values within the TechDB.



**New Options for Defining Cut Amount Parameters under Contour Tab in Operations Parameters Form within TechDB User Interface**





**New - Optimized Pecking Option for Drill Toolpaths generated for Condensed Hole Features**

**Purpose:**

To provide the **Optimized Pecking** option for Drill operations generated for *Condensed Hole* features so that peck moves can be eliminated in the empty space(s) where a hole feature intersects with other features

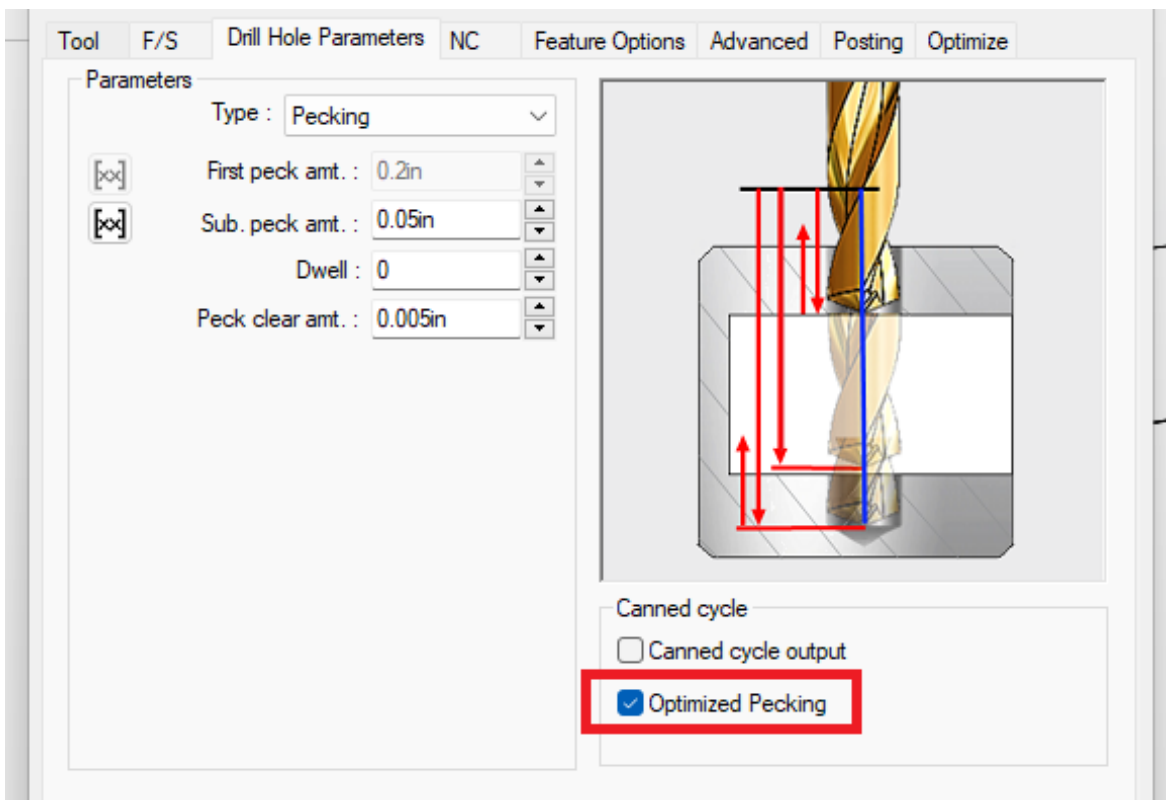
**Implementation:**

Condensed hole features consist of two or more coaxial condensed cylinders of identical diameter. These cylinders are either directly separated by an empty space or by an intersecting Mill feature that cause the cylinders to remain separated.

In previous versions of *CAMWorks*, when Drill toolpaths were generated for Condensed Hole features, the toolpaths would contain peck moves even in the locations where hole features intersected with other features.

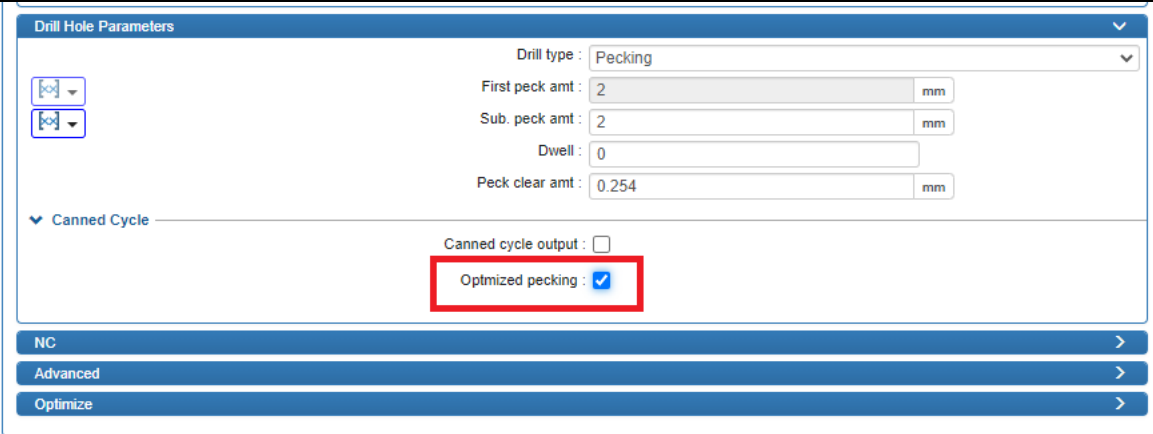
From *CAMWorks 2024 BETA* version onwards, an option named **Optimized Pecking** will be available under the **Drill Hole Parameters** tab of *Drill* operations.

This parameter will be enabled when the **Type** is set to *Pecking*, *High Speed Pecking* or *Variable Pecking* and the **Canned cycle output** checkbox option is unchecked. The settings associated with this checkbox option will only affect condensed hole features (both automatically recognized and interactively inserted condensed hole features.)



**New 'Optimized Pecking' checkbox option under Drill Hole Parameters Tab for Drill Hole Operations**

- When the **Optimized Pecking** checkbox option is unchecked, the drilling toolpath will be generated for the combined depth of the condensed holes. Under default settings, this checkbox remains unchecked. These default settings can be changed in **Drill Hole Parameters** tab under **Drill form** within the TechDB.



**Optimized Pecking Option under Drill Hole Parameters tab in Drill Hole Form of TechDB**

- When the **Optimized Pecking** checkbox option is checked, the drilling toolpath will be modified such that after machining the first cylinder in the group, the tool will directly move to the top of the succeeding cylinder and will start the peck moves from the distance set as Peck clearance amount. The feedrate assigned to this move will be the same as the Z feedrate. There will be no peck moves between the two cylinders.

<p><b>Drill Toolpath when 'Optimized Pecking' option is unchecked:</b> Observe that peck moves are created in the empty space.</p>	<p><b>Drill Toolpath when 'Optimized Pecking' option is checked:</b> Observe that there are no peck moves in the empty space. The tool will directly move the start point/Peck Clearance Amount point of the next cylinder.</p>



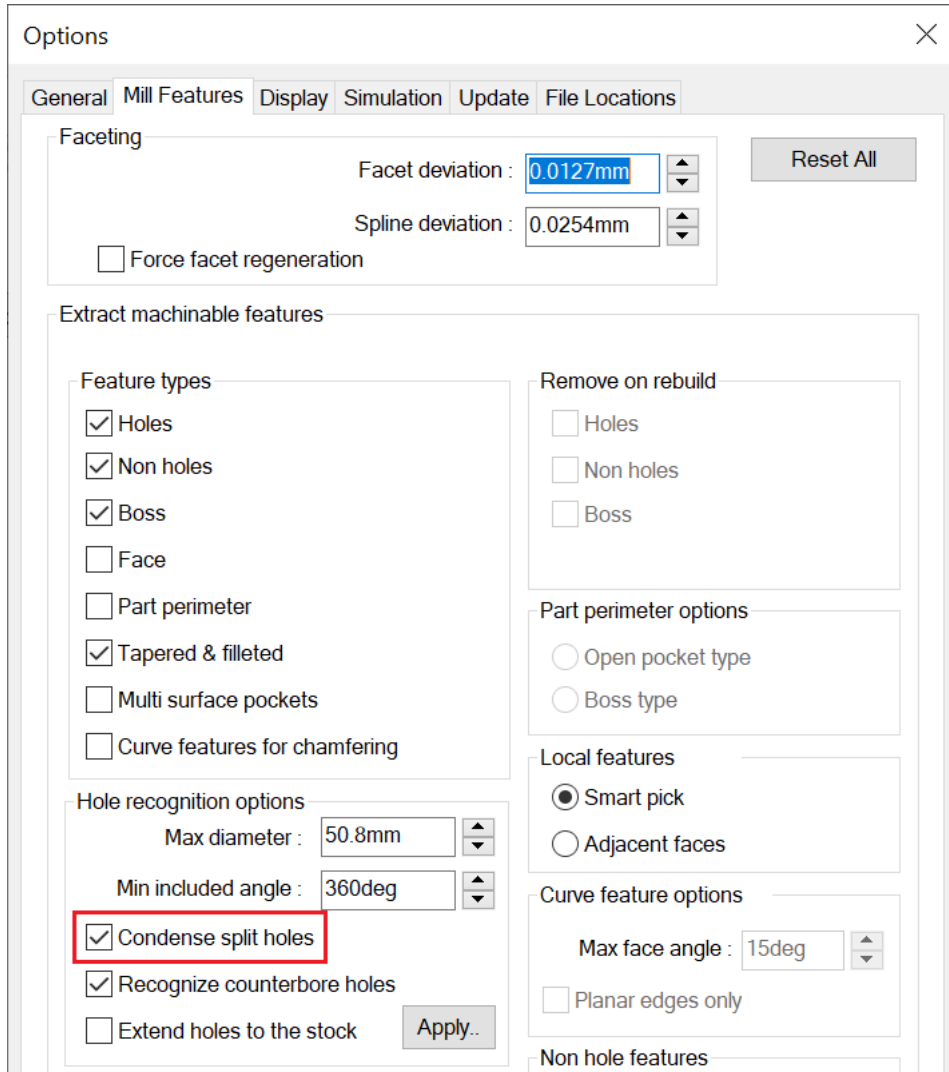
**Improved - Improved Options for Condensed Hole Features**

**Purpose:**

To provide the ability to interactively split/condense hole features and display the number of holes/cylinders comprising condensed hole features as well as parameters associated with each such component hole/cylinder

**Implementation:**

For simple holes, Counterbore holes and Countersink holes, existing versions of CAMWorks provide the **Condense split holes** option. This option is available under the **Mill Features** tab of the **CAMWorks Options** dialog box.



**Mill Features tab under CAMWorks Options Dialog Box**

When this checkbox option is checked and *Automatic Feature Recognition* is executed, hole features which are co-axial and of same diameter are condensed into a single hole feature. This condensed hole will be represented by a single hole feature node in the Feature tree.

**Improved: Displaying the Number of Cylinders present for Each Condensed Hole Feature Node**

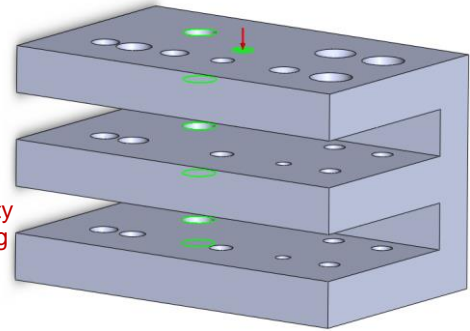
From *CAMWorks 2024 BETA* version onwards, the display of all condensed hole feature nodes in the Feature tree has been enhanced to indicate the number of condensed cylindrical faces the comprise such a hole.



# What's New in CAMWorks 2024 BETA



Number indicating the quantity of cylindrical faces comprising the condensed hole.

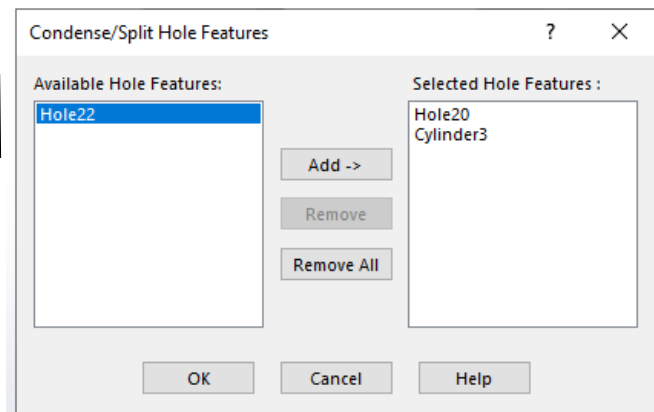
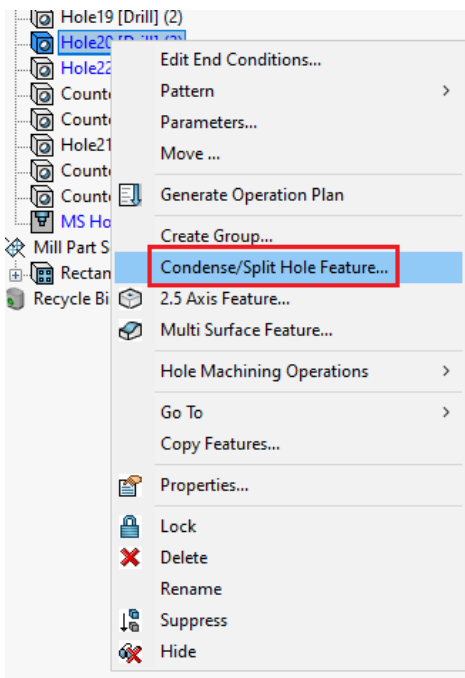


**Condensed Hole Feature in Feature Tree tab**

## Improved: Option to interactively Condense/Split Hole Features

If the **Condense split holes** option under the *CAMWorks Options* dialog box is left unchecked and *Automatic Feature Recognition* was executed, then all co-axial hole features will be listed as individual features under the different Mill Part Setups in the Feature tree. In previous versions of *CAMWorks*, there was no option to interactively condense such co-axial holes. Also, no option was available to split condensed hole features.

From *CAMWorks 2024 BETA*, an option to interactively condense/ split hole features has been introduced. This option of **Condense/ Split Hole Feature** will be available in the context menu of hole features listed in the Feature tree.



**New Option 'Condense/Split Hole Feature...' in Context Menu for Hole feature Nodes in Feature Tree**

**Condense/Split Hole Features Dialog Box displayed on executing the 'Condense/Split Hole Feature...' option**

Executing the **Condense/ Split Hole Feature** command will display the **Condense/Split Hole Features** dialog box. The parameters within this dialog box can be used to:

- Condense one or more eligible hole features into a single hole feature.  
(Use the **Add** button to select hole features listed in the **Available Hole Features** list box and then click the OK button.)



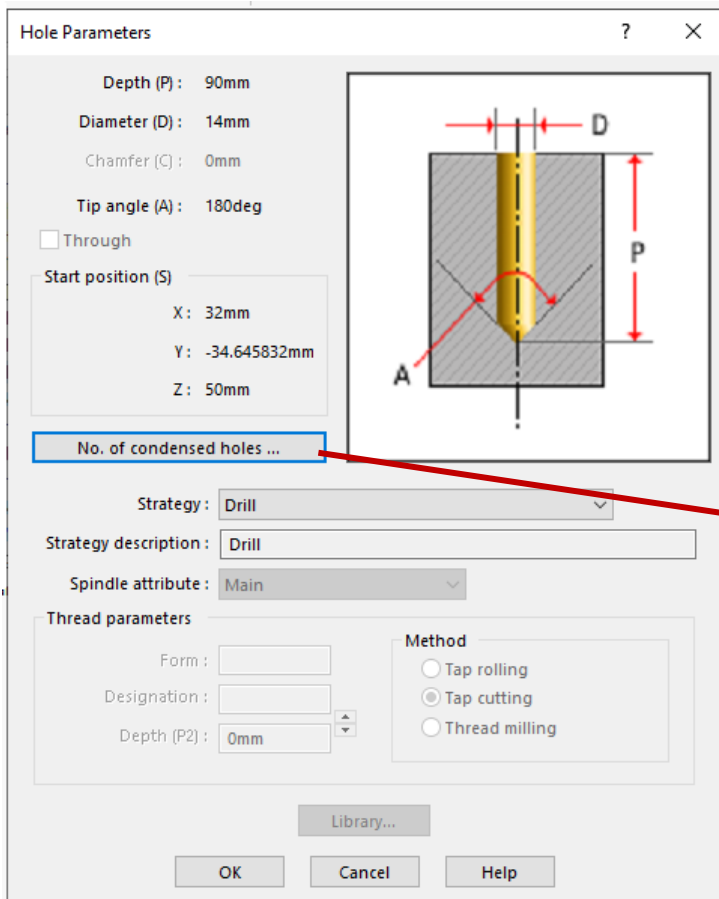
- Split one or more hole features constituting a condensed hole feature into separate hole features.

(Select the hole features to be removed from the **Selected Hole Features** list box and click the **Remove** button to separate it from the condensed hole feature. When you click the OK button, the selected features will be listed as separate hole features in the Feature tree.)

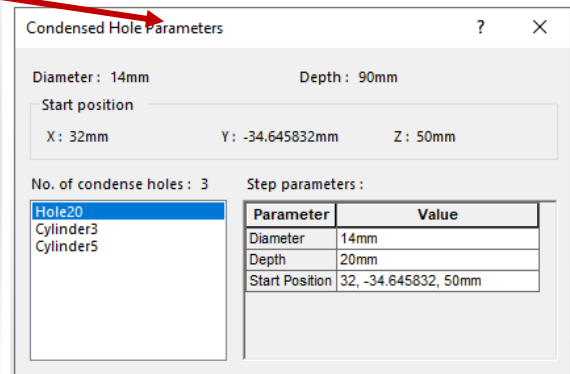
**Improved: Viewing the Number of Condensed Holes in Hole Parameters Dialog Box**

In previous versions of CAMWorks, the **Hole Parameters** dialog box had neither the mechanism to indicate the number of cylinders comprising a condensed hole nor their associated *parameters*.

From CAMWorks 2024 BETA version onwards, a new button named **Number of Condensed Holes...** has been introduced in this dialog box. This button will be enabled only for condensed hole features and disabled for other hole feature types. Clicking on this button will display the **Condensed Hole Parameters** dialog box. All the cylinders/holes comprising the condensed hole will be displayed in a list within this dialog box. Selecting any cylinder/hole in this list will display its associated parameters. The contents displayed within this dialog box are read-only.



**Hole Parameters Dialog Box**



**Condensed Hole Parameters Dialog Box**



**Improved - Dynamic Highlighting of Nodes in Feature Tree and Operation Tree**

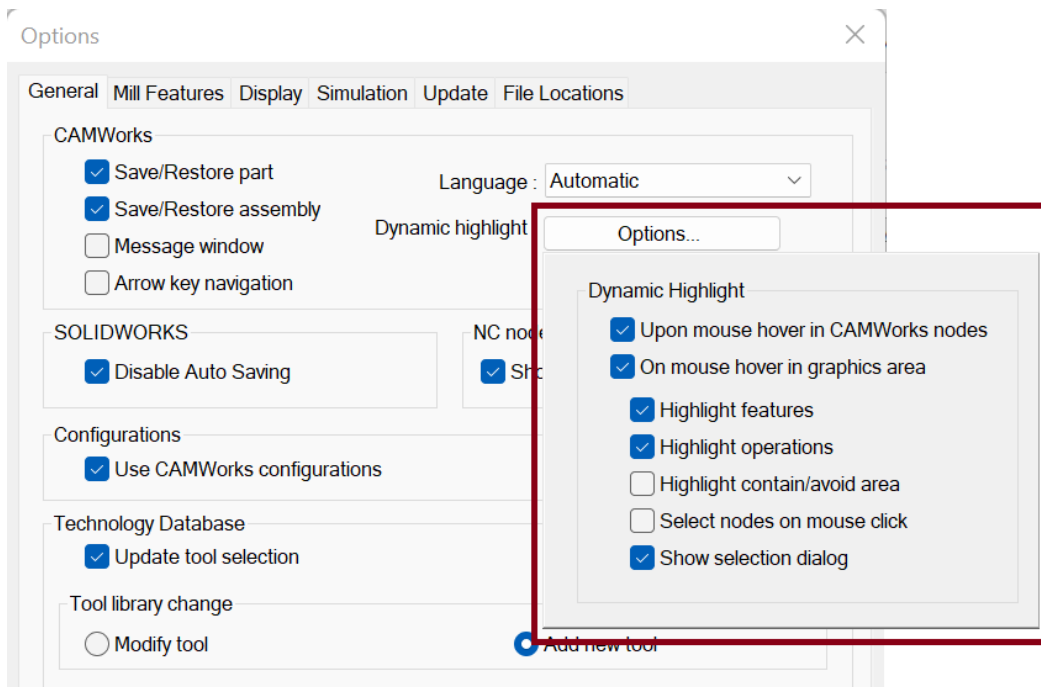
**Purpose:**

To provide the option of identifying the node(s) associated with one or more entities of the part model/assembly that are selected in the graphics area

**Implementation:**

In previous versions of CAMWorks, the checkbox option of **Dynamic Highlights** was available under **General tab** of **CAMWorks Options** dialog box.

From CAMWorks 2024 BETA version onwards, the **Dynamic Highlights** functionality has been enhanced to additionally provide the reciprocal option viz. the option of identifying the node(s) associated with one or more entities of the part model/assembly that are selected in the graphics area. The **Dynamic Highlights** checkbox has been replaced with the **Dynamic Highlights Option...** button.



**Dynamic Highlight Options available under General Tab of CAMWorks Options Dialog Box**

Clicking on this button displays a dropdown list of available highlighting options. Users can enable/disable these display options as per their requirements. Following are the options:

- **Upon mouse hover in nodes**

This checkbox option represents the existing functionality of dynamic highlighting.

- **On mouse hover in graphics area**

This checkbox has five sub-options associated with highlighting nodes in the Feature tree/ Operation tree when the entities associated with them are highlighted/selected on the part model/assembly in the graphics area. Selecting one or more of these sub-options will automatically place a check in this checkbox.

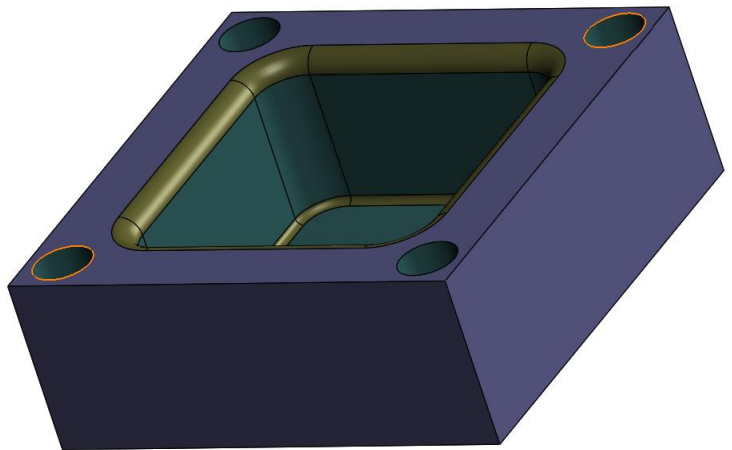
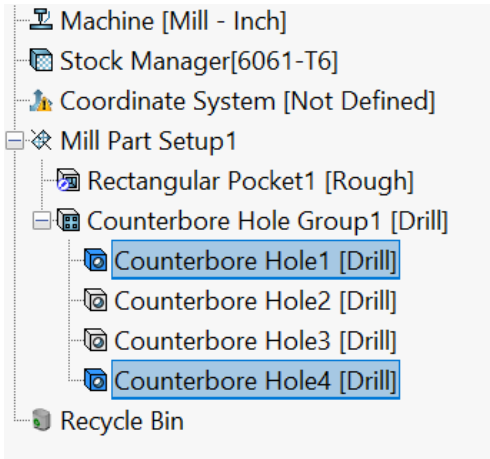
- **Highlight features**

If this checkbox option is checked, it will highlight feature nodes in the Feature tree when one or more SOLIDWORKS entities (face, edge, sketch, etc.) of the part model are selected in the graphics area. This selection can be in form of mouse hover over a single SOLIDWORKS entity or manual selection of SOLIDWORKS entities. The feature nodes highlighted in the Feature tree will be those feature(s) involving or associated with the selected SOLIDWORKS entities.

In Assembly mode, when SOLIDWORKS entities are selected or mouse hovered in the graphics area for any part instance, then the corresponding SOLIDWORKS entities for the



seed part and all other part instances too will be highlighted. Under the **Part Manager** item in the Feature tree, the Feature nodes associated with the selected/mouse hovered entities will be highlighted for both the seed part and all other part instances. The applicable feature nodes under the Assembly Setup item in the Feature tree will also be highlighted.



**Sample Image of Features nodes being highlighted in Feature tree when a SOLIDWORKS entities on the part model are hovered over/manually selected**

- **Highlight operations**

If this checkbox option is checked, it will highlight those Operation nodes in the Operation tree which have toolpaths for the selected SOLIDWORKS entities in the graphics area. This selection can be in form of mouse hover over a single SOLIDWORKS entity or manual selection of one or more SOLIDWORKS entities.

- **Highlight contain/avoid area**

If this option is checked, it will enable users to visually identify operations for which Contain Areas and/or Avoid Areas have been defined.

- **Select nodes on mouse click**

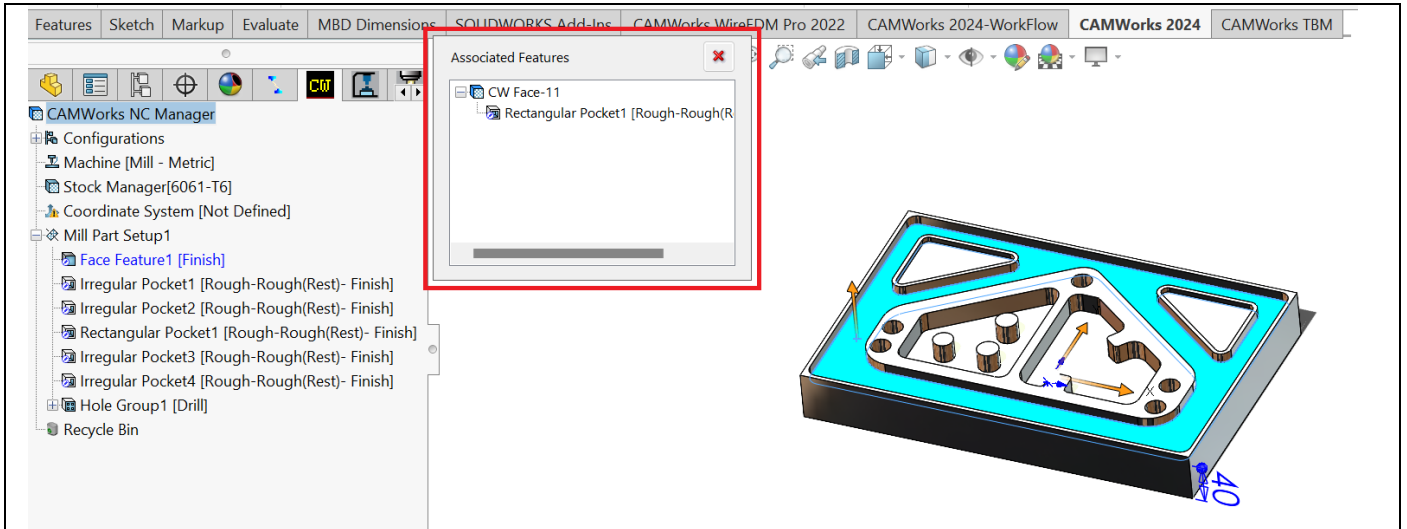
If user selects one or more faces on the part model or mouse hovers on any face of the part model/assembly in the graphics area, then the corresponding features or operations in the active tree will be selected and highlighted.

- **Show selection dialog**

Checking this option will enable the display of the **Associated Features** dialog box and **Associated Operations** dialog box. These dialog boxes list the names of associated features and operations, respectively.

**Associated Features Dialog Box**

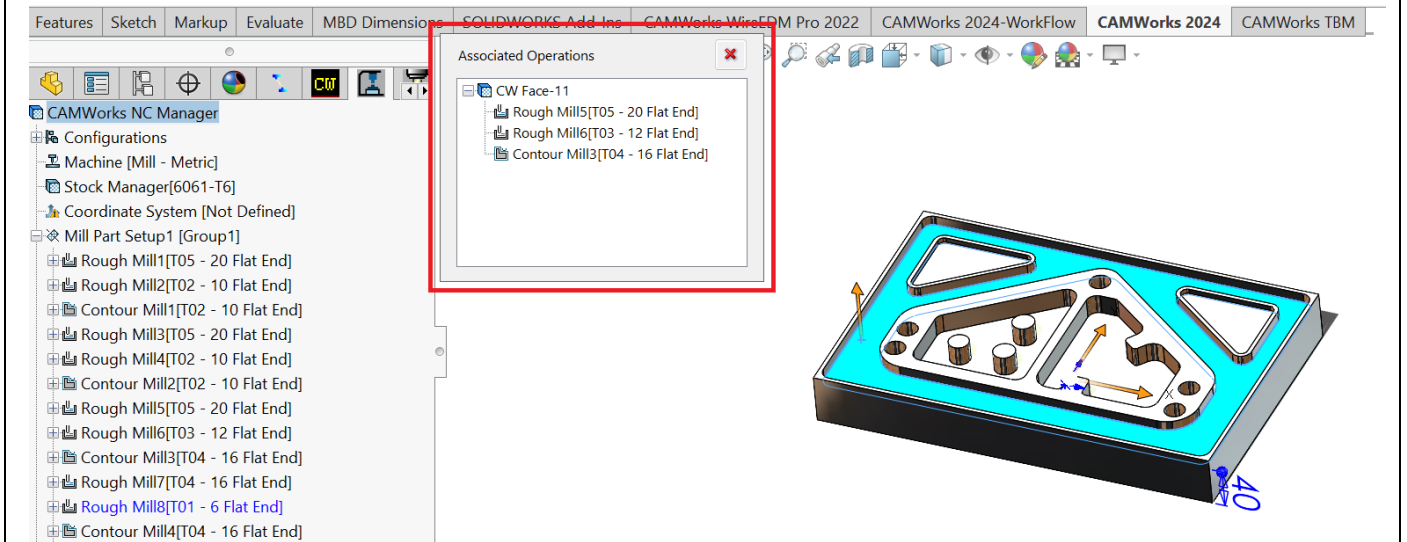
Within the *Associated Features* dialog box, the listed feature nodes will get dynamically updated as you mouse hover over other faces/edges or manually select other faces/edges of the part model/assembly. If you click on any feature node displayed within this dialog box, then its corresponding entry will be highlighted in the *Feature Tree*.



**'Associated Features' Dialog Box Displayed when a faces/edges on the part model/assembly are mouse hovered or manually selected**

**Associated Operations Dialog Box**

Within this dialog box, the associated operations will usually be operations used to machine the selected faces and/or edges. However, if any selected face has a *Contain Area/Avoid area* defined or if any selected sketch has been used to define a *Contain Area/Avoid area*, then the associated operation will be the *Operation* node under which that specific *Contain Area/ Avoid area* has been defined.



**'Associated Operations' Dialog Box Displayed when a faces/ edges on the part model/assembly are mouse hovered or manually selected**





**Improved - Improved Leadin/Leadout for Thread Milling Toolpaths for Hole Features**

**Purpose:**

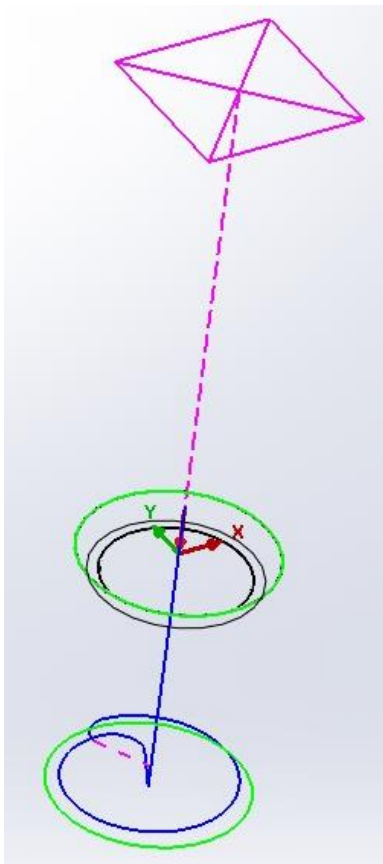
**Implementation:**

For Thread Milling toolpaths, the Leadin and Leadout can be defined using user-defined parameters.

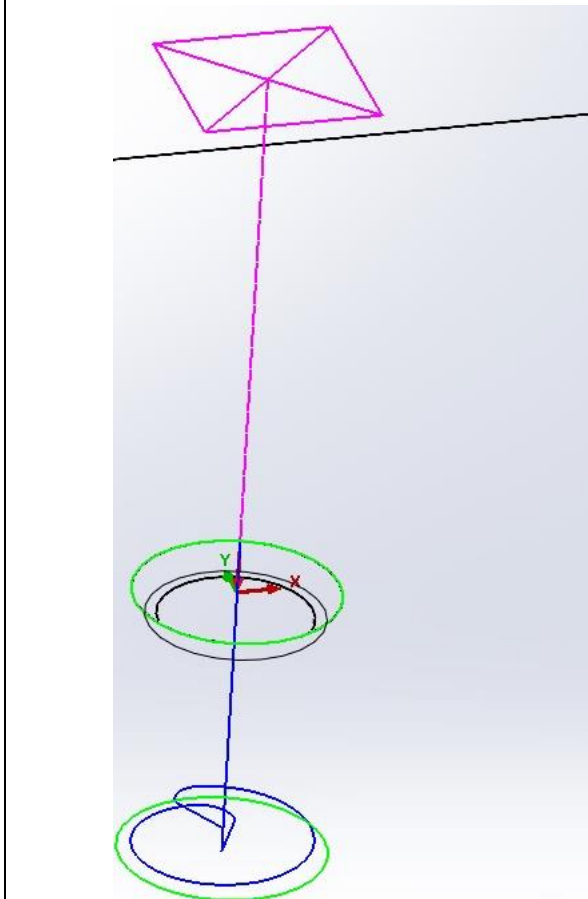
In previous versions of CAMWorks, the leadin/leadout moves for Thread Milling toolpaths generated using Multi-point Thread Mill tools were occasionally unsatisfactory due to incorrect generation of rapid moves and feed moves causing the Cutter compensation to be at the wrong places.

From **CAMWorks 2024 BETA** version onwards, improvements have implemented for Leadin/Leadout moves associated with Thread Milling toolpaths generated for hole features as follows:

1. If the leadin and/or Leadout amount is set to zero, the move from center of the hole feature to the start of the cut move will be a feed move. This will ensure that the CNC compensation is applied on this linear move.
2. If the leadin and leadout amount is a non-zero value, then the move from center of the hole feature to the start of leadin move and the move from end of the lead out move to the center of the hole feature will be a rapid move.
3. The same above-mentioned behavior will be observed in all the passes of the Thread Mill toolpath.
4. The Leadin /Leadout moves will be independent of the **Gouge check** checkbox option's status.



Sample Toolpath in Previous CAMWorks Version



Same Sample Toolpath in CAMWorks 2024 Version



# Turn / Mill-Turn

## **New - Options to Highlight Recognized/Unrecognized Faces of the Turn Part Models**

**Purpose:**

- To provide the option to highlight faces on the Turn part model in the graphics area that have recognized by either **Automatic Feature Recognition** or **Interactive Feature Recognition** (both *Plane section* and *Revolved section*)
- To provide the option to highlight faces on the Turn part model in the graphics area that have not been recognized by either **Automatic Feature Recognition** or **Interactive Feature Recognition**.

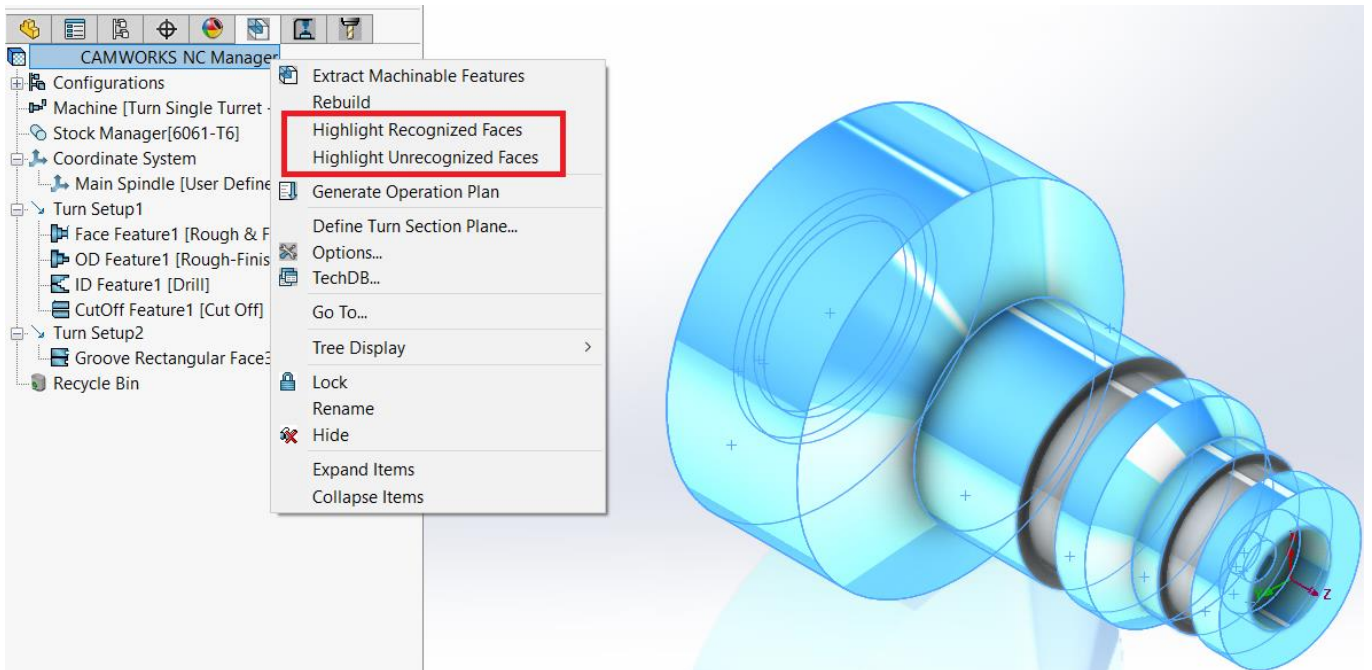
**Implementation:**

In previous versions of CAMWorks, the option to highlight faces on the active Mill part model that have been recognized using *Automatic Feature Recognition* was available in the form of **Highlight Recognized Features** option in the context menu of CAMWorks NC Manager node in the Feature tree. Similarly, the option to highlight faces left unrecognized by *Automatic Feature Recognition* was available in the form of **Highlight Unrecognized Features** option in the context menu of CAMWorks NC Manager node in the Feature tree.

From CAMWorks 2024 BETA version onwards, this functionality has been extended to Turn Part models. This functionality has been additionally enhanced to recognize faces that have been recognized by *Automatic Feature Recognition* as well as *Interactive Feature Recognition* (both *Plane section* and *Revolved section*). It will be available as context menu options of **Highlight Recognized Features** and **Highlight Unrecognized Features** for the CAMWorks NC Manager node in the Feature tree.

**Note:**

- This functionality to highlight faces recognized or left unrecognized only in Turn Part mode. It will not be available in Turn Assembly mode.
- To use this functionality for Turn legacy parts, you need to execute the **Rebuild** command to rebuild the CAM data. Once rebuilt, the **Highlight Recognized Features** and **Highlight Unrecognized Features** command will become available in the CAMWorks NC Manager node's context menu.



**New Options in Context menu of CAMWORKS NC Manager Node in Feature tree for Turn Mode**



**New - Defining SOLIDWORKS Fixture Coordinate System (FCS) in Turn Assembly Mode**

**Purpose:**

To provide the ability to define **Fixture Coordinate Systems** for Turn Assemblies in CAMWorks Turn mode

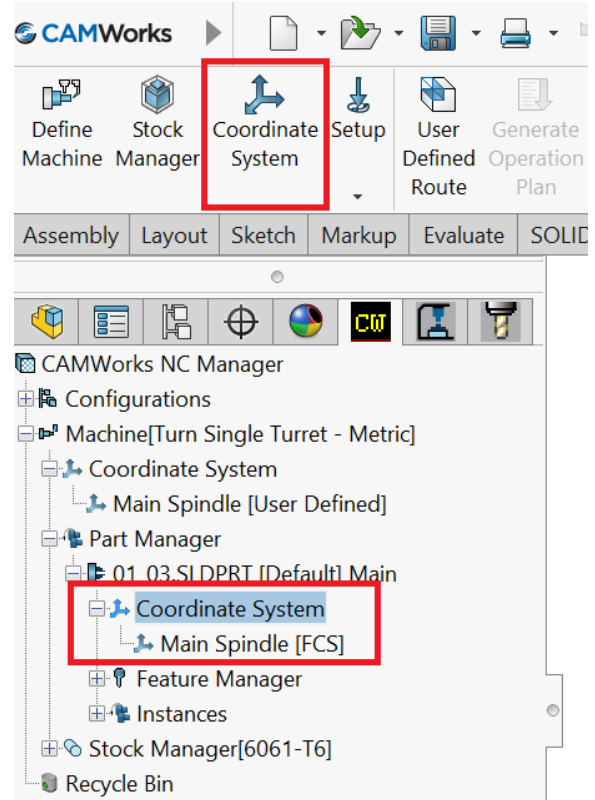
**Implementation:**

In previous versions of CAMWorks, the option to define **SOLIDWORKS Fixture Coordinate Systems** was available only in *Mill Assembly* mode. From CAMWorks 2024 BETA version onwards, this functionality has been extended to Turn Assembly mode.

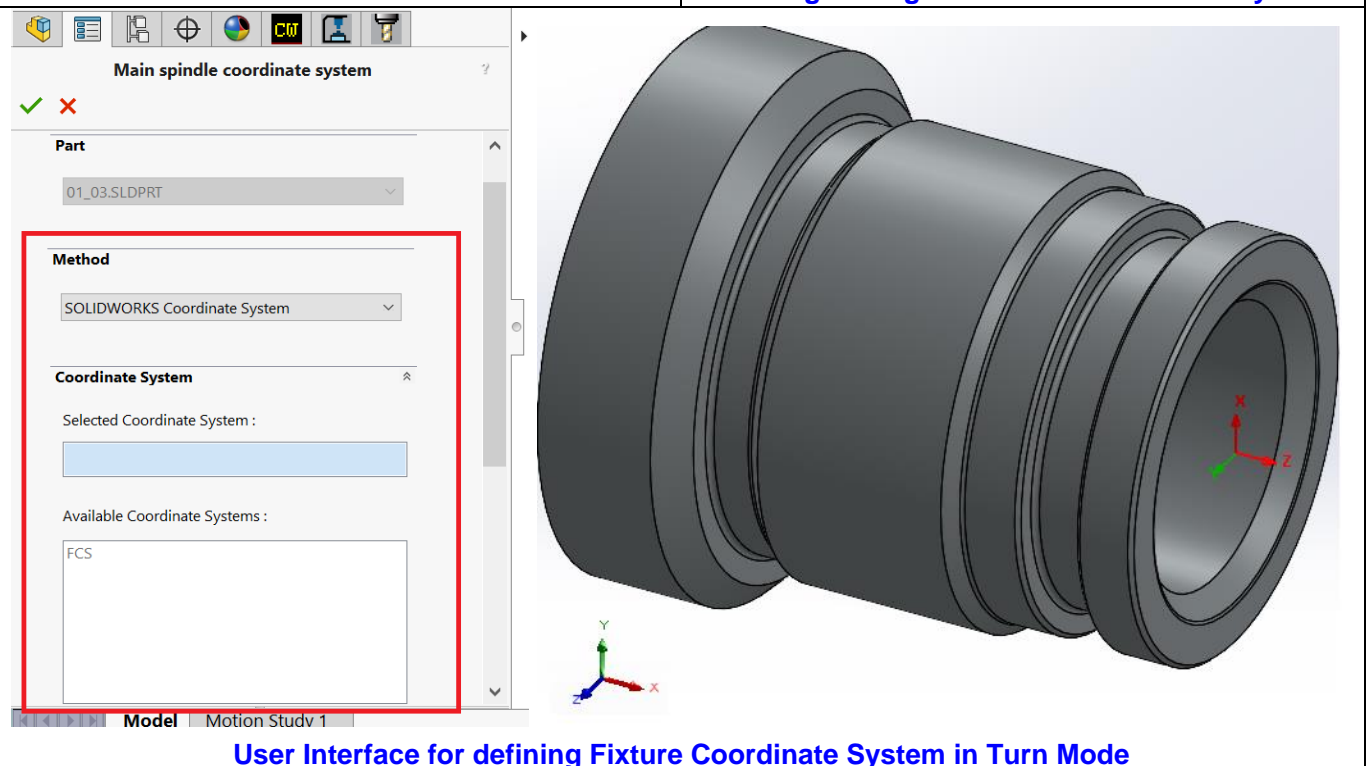
While defining the Coordinate system for Machine and Parts in the Assembly mode, users can now select the available SOLIDWORKS Coordinate Systems.

The option to select **SOLIDWORKS Coordinate Systems** will be available within the user interface for defining/editing the coordinate system.

The existing rules of defining different coordinate systems for the spindle will continue to apply as they are in the current form.



**Commands to invoke the User Interface for defining/editing the Fixture Coordinate System**



**User Interface for defining Fixture Coordinate System in Turn Mode**



**New - Option to Create Centralized Library of Turn Tool Blocks in TechDB**

**Purpose:**

To provide the option to create a centralized library of Turn tool blocks within the Technology Database so that the Tool block selection process when defining tools in TechDB gets automated.

**Implementation:**

For Turn and Mill-Turn machine simulation using the CAMWorks Virtual Machine application, users need to assign tool block for the Turn tools.

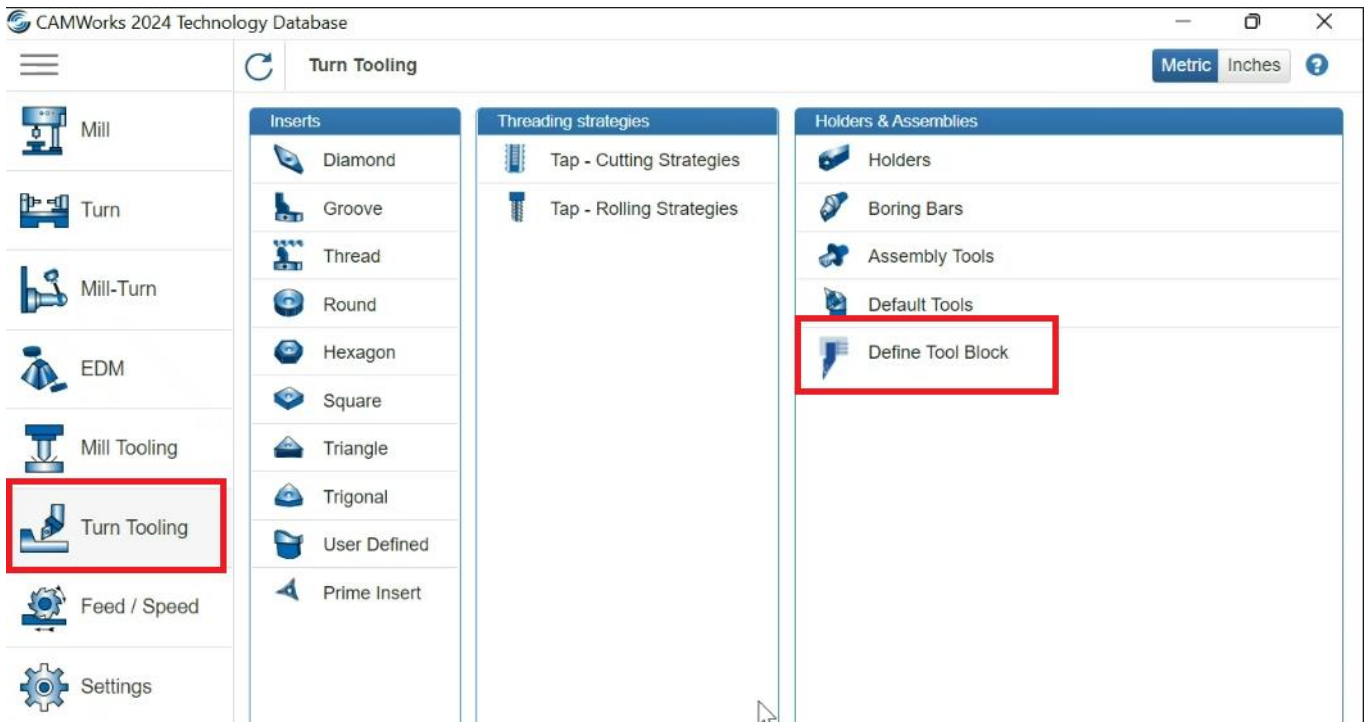
In previous versions of CAMWorks, the tool block for any Turn tool could be assigned only in the **Operation Parameters** dialog box. All Turn Tool Blocks associated with a specific Turn/Mill-Turn machine could be created and stored in an assigned folder. If the tool blocks had to be reused for other Turn/Mill-Turn machines, then users had to manually replicate the requisite tool block files in other folders associated with those respective machines.

From CAMWorks 2024 BETA version onwards, the option to create a centralized library of Turn tool blocks within the Technology Database will be available. These tool blocks saved in the TechDB can then be assigned to Turn tools in the Tool crib assigned to different machines. It will thus aid in automating the Tool block selection process when defining tools in TechDB.

This new functionality will replace the previously cumbersome method of defining tool blocks for each Turn/Mill-Turn machine.

**Menu Option:**

The menu option labeled **Tool Blocks** for invoking the user interface associated with Tool Blocks will be available under **Holders & Assemblies** sub-menu of the **Turn Tooling** menu in the TechDB.



**'Define Tool Block' Menu Option under Turn Tooling Menu of Technology Database**




**Tool Blocks User Interface:**

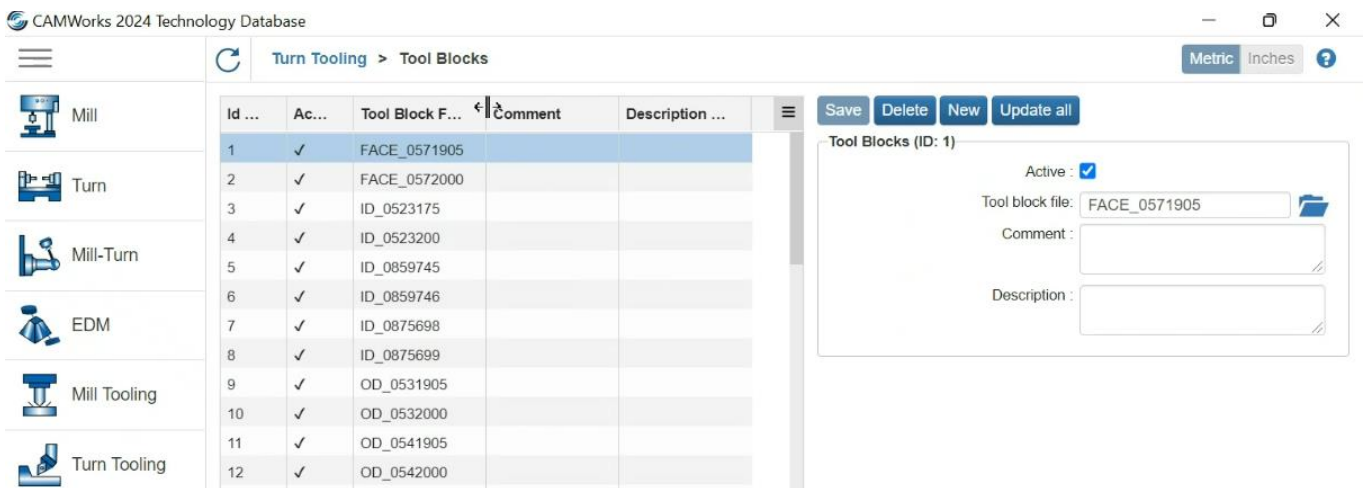
The **Tool Blocks** user interface is displayed when you click on the **Tool Blocks** menu option.

**Table of Tool Blocks**

All the Tool Blocks defined and saved within the TechDB on the left hand side of the user interface in a tabular format. This table is populated by the available \*.TBDINF files present in the folders and respective sub-folders of the user defined path of CAMWorks Virtual machines linked to CNC machines defined in the TechDB.

**Tool Blocks Form**

Use the **Tool Blocks** form of the right hand side of the user interface to edit any saved Tool Block entry or add a new Tool Block entry. To add a new tool block entry, use the Browse button  adjacent to this field in order to select the desired Tool Block Info file (\*.TBDINF). Once selected, the name of the selected file is displayed in this field. When you mouse hover over the Tool block file field, the file path to that specific \*.TBDINF file will be displayed as a screen tip.




Turn Tool Blocks User Interface

**Assigning Tool Blocks to Turn Tool Cribs in TechDB**

In the Turn Tool Cribs user interface of TechDB, the option to assign Tool Blocks to Turn tools is available in the form of the newly available **Tool Block** parameter. Use this parameter to browse, select and assign the desired tool block for the current station of the active tool crib. When no tool block is selected, a value of '-1' will be displayed in this parametric field.

**Steps to assign a tool block using the Tool Block Parameter**

1. Click on the **Browse tool block file** button  adjacent to this parametric field.
2. Clicking this command button will invoke the Tool Blocks pop-up user interface. To select a tool block, highlight that tool block in the table of Tool Blocks on the left hand side of this user interface.
3. Click the **Select** button at the top left corner. The pop-up user interface will close and revert to the previous user interface viz. the **Tool Cribs** user interface. The ID associated with the selected Tool block will be displayed in the **Tool Block** parametric field. This indicates that the selected tool block has been assigned for the current station of the active tool crib.
4. The name of the selected **Tool block** file will be displayed in the **Name** field below the **Tool Block** parameter within the **Tools** form.



**Turn Tool Cribs User Interface**

I...	Stn. No. ...	Sub. No....	Station I D	Comb I D ...	Tool Type	Tool I D...	Holder
14	1	0		N.A.	Insert-Holder	12	Down Lef
15	2	0		N.A.	Insert-Holder	14	Left Up
16	3	0		N.A.	Insert-Holder	13	Down Lef
17	4	0		N.A.	Insert-Holder	15	Down Lef
18	6	0		N.A.	Center Drill	3	Left Up
19	9	0		N.A.	Insert-Holder	18	Down Lef
33	10	0		N.A.	Insert-Holder	38	Down Lef
34	11	0		N.A.	Insert-Holder	43	Down Lef

**Tools (ID: 14)**

Stn. No. : 1  
 Sub. No. : 0  
 Station ID :  
 Comb ID : N.A.  
 Type of Tool : Insert-Holder  
 Tool ID : 12  
 Holder Orientation : Down Left  
 Auto tip offset :   
 Z Offset : 0 mm  
 X Offset : 0 mm  
 Gage X offset : 46 mm  
 Gage Y offset : 0 mm  
 Gage Z offset : 17 mm  
 Key Parameters : CNMG 431 80DEG SQR HOLDER  
 Description : None

**Holder ID** : -1  
**Tool Blocks** : 76  
**Name** : OD\_NKM0162000

Always load in turret :

Turn Tool Cribs User Interface

**User Interface displayed on clicking the *Browse tool block file* button adjacent to Tool Block field**

Id ...	Active ...	Tool Block File	Comment	Description ...
60	✓	OD_0532000		
61	✓	OD_0541905		
62	✓	OD_0542000		
63	✓	OD_8694753		
64	✓	OD_BGL03175		
65	✓	OD_BGL03200		
66	✓	FACE_NKM0021620		
67	✓	FACE_NKM0181905		
68	✓	FACE_NKM0182000		
69	✓	ID_NKM0113221		

**Tool Blocks (ID: 76)**

Active :   
 Tool block file : OD\_NKM0162000  
 Comment :  
 Description :

User Interface displayed on clicking the *Browse tool block file* button adjacent to Tool Block field

**How the 'Tool Block' Parameter Affects the CAMWorks Application**

Within the CAMWorks application, under **Station** page of **Tool** tab for **Operation Parameters** dialog box of **Turn** operations, the tool block assigned for the current station of the active tool crib is displayed. If the **Always load in Turret** checkbox option is checked, then the tool will be loaded at a specific station of the tool crib. The name of the **Tool Block** assigned to that specific station will be displayed within the **Block Name** column within the **Virtual Machine Tool Block** group box under **Station** page of the **Tool** tab.



**New - Option to start programming of Turn/Mill-Turn parts by holding them in Sub Spindle**

**Purpose:**

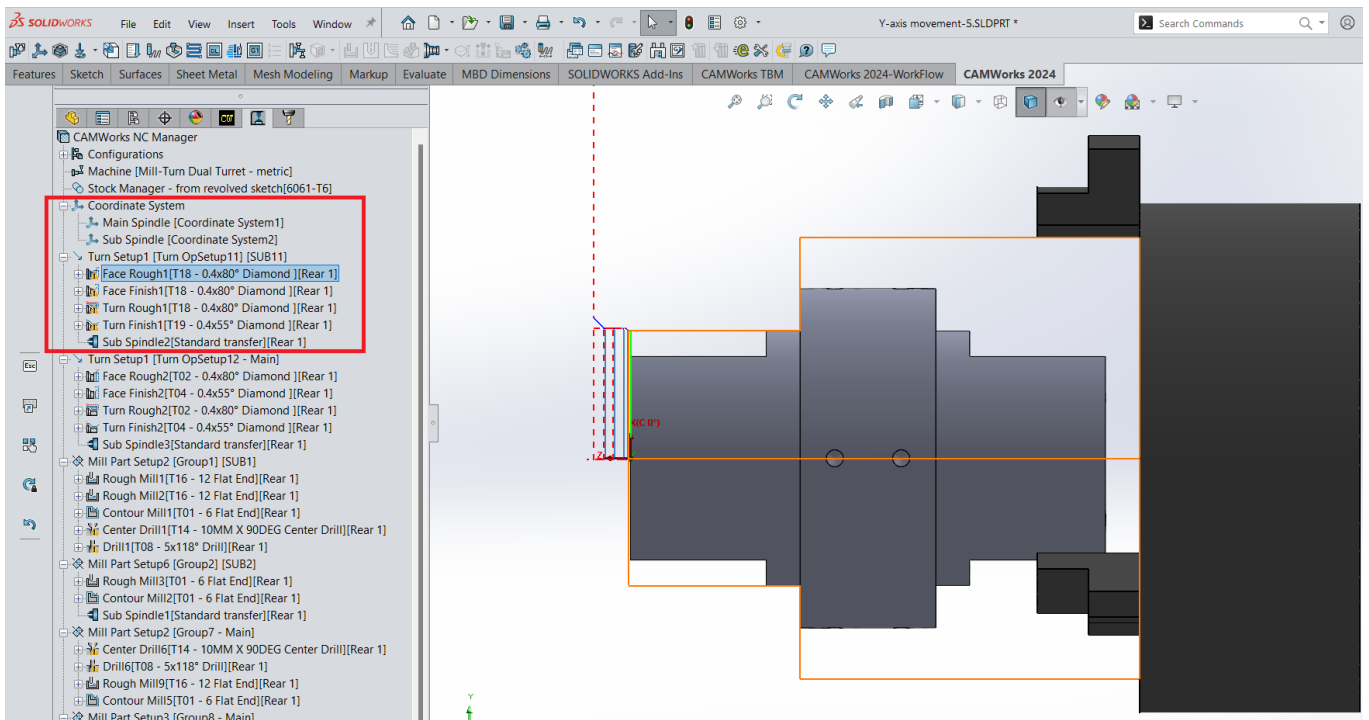
To provide the option to program Turn and Mill-Turn parts held in the Sub Spindle instead of Main Spindle at the start of the machining cycle

**Implementation:**

In previous versions of CAMWorks, for dual spindle Turn and Mill-Turn, it was always assumed that a Turn/Mill-Turn part to be programmed using CAMWorks will always be held in Main Spindle and then transferred to the Sub Spindle for subsequent operations. However, at times, there will be cases where users might want to start machining a part by holding it in the Sub Spindle. This option was not available.

From CAMWorks 2024 onwards, the functionality to begin programming Turn/Mill-Turn parts held in the Sub Spindle will be available. If any part is held in the Sub Spindle and this is visually indicated in the Operation tree (via the positioning of the corresponding node at the top of the Operation tree), then it will be assumed that the Sub Spindle is at its home position. The toolpaths and NC code will be generated accordingly.

The existing functionality to transfer the part between Sub Spindle and Main Spindle and vice versa will be available.



**Sample Image of a Turn Part Model held initially in Sub Spindle. Observe that the first Turn Setup in the Operation Tree is from the Sub Spindle.**



# Multiaxis

## New - Tilt Tool by Contact Point option for Multiaxis Mill Operations

### Purpose:

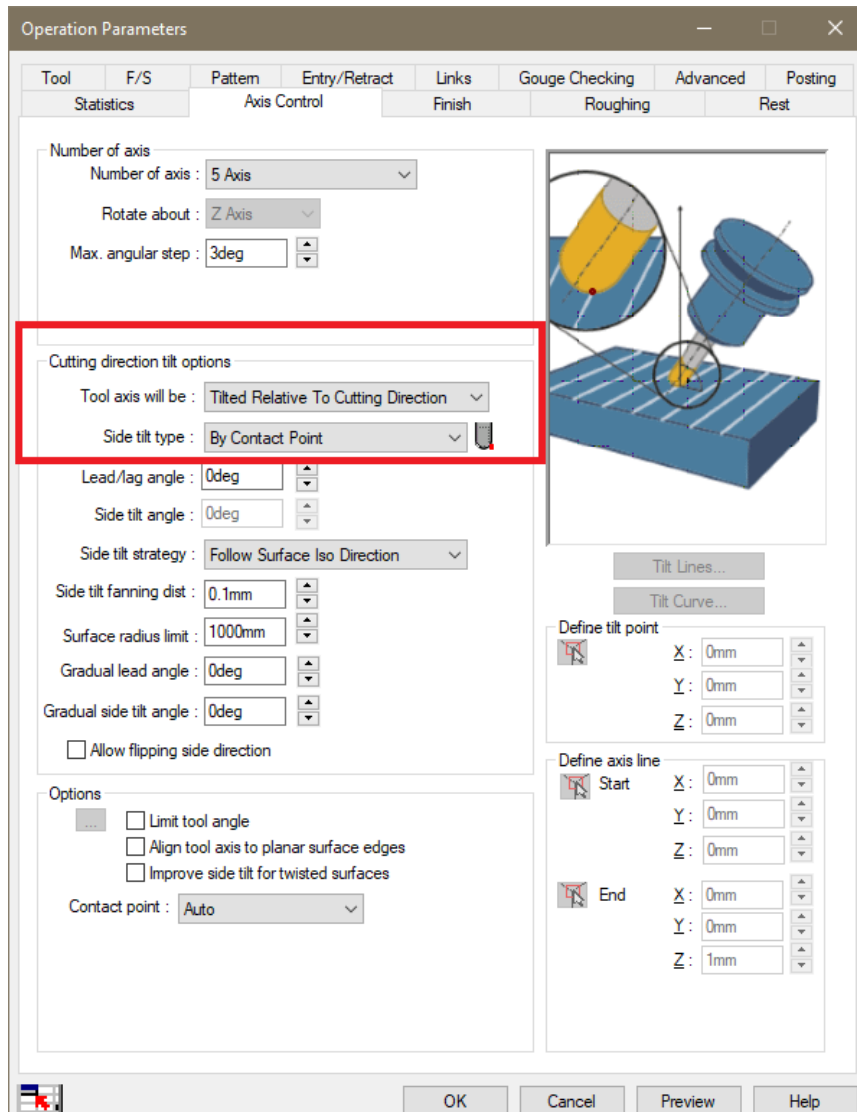
Option to tilt tool by specifying a contact point for the cutting direction for Multiaxis Mill operations

### Implementation:

For Multiaxis Mill operations, the **Axis Control** tab under the **Operation Parameters** dialog box allows you to define the tool orientation relative to the surface normal.

In previous versions of CAMWorks, within the **Cutting direction tilt options** group box the **Axis Control** tab, when the **Tool axis will be** parameter was set to **Tilted Relative to Cutting Direction**, the **Tilt tool By Angle** strategy was employed. The angle specified in the **Side Tilt Angle** field defined the side tilt value of the tool axis from the surface normal direction based on the toolpath cut direction.

From **CAMWorks 2024 BETA** version onwards, an additional tool tilt strategy will be available when the **Tool axis will be** parameter is set to **Tilted Relative to Cutting Direction**. To accommodate this new option, a dropdown list labeled **Side Tilt Type** will be displayed below the **Tool axis will be** parameter. This dropdown list will be displayed only when **Tool axis will be** parameter is set to **Tilted Relative to Cutting Direction**.



New 'Side Tilt Type' dropdown list under Axis Tab for Multiaxis Mill Operations






Following are the two options that will be available within the Side Tilt Type dropdown list:

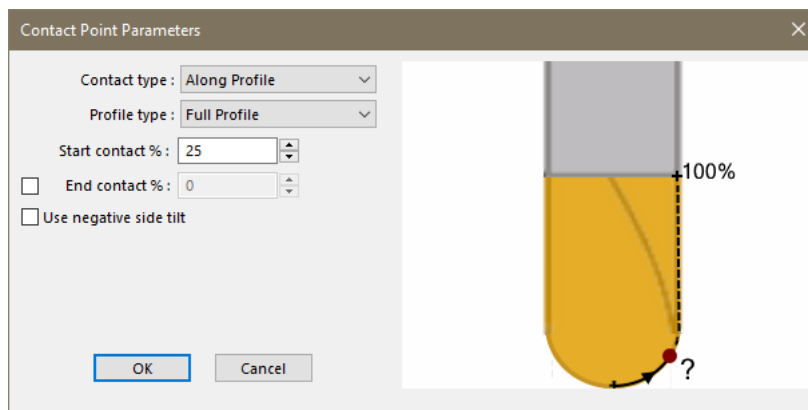
- **By Angle**

This is the existing strategy that was available in previous versions of CAMWorks too as explained in the previous paragraph. From *CAMWorks 2024 BETA* version onwards, it will be available only when the **By Angle** option is selected. Use the **Side tilt angle** parameter to specify the tilt angle.

- **By Contact Point**

This is the new tilt tool option introduced from *CAMWorks 2024 BETA* version onwards. Set the **Side tilt type** to **By Contact Point** if you wish to specify the tool contact point using contact point parameters. When this option is selected, the **Contact Point** command button  will be displayed adjacent to the *Side tilt type* dropdown list.

Clicking on this button command will display the **Contact Point Parameters** dialog box. Use this dialog box to define the contact point parameters for the cutting direction.



**Contact Point Parameters Dialog Box**

**Parameters in the Contact Point Parameters Dialog Box**

**Contact Type**

The contact point can be defined in two ways:

- **By Height:** When this option is selected, the contact point is defined by the distance from the lowest point of the selected profile section to the contact point in the tool axis direction. The distance is percentage of the length of the selected profile section.
- **Along Profile:** When this option is selected, the contact point is defined by the distance from the lowest point of the selected profile section to the contact point along the tool profile.

**Profile Type:** When defining the contact point, you can specify a profile type that identifies the following different areas of the tool that are to be considered using the Profile Type dropdown list. The Profile type dropdown list will display only those profile sections of the tool that are available on the selected tool assigned to the operation.

**Start Contact %:** Use this parameter to define the start contact point as a percentage of the height or along the profile. Allowed values are in the range of 0% to 100%.

**End Contact % (Checkbox option and associated field):** The **End Contact%** field will be enabled when you place a check in the **End Contact %** checkbox. Use this parameter to define the end contact point as a percentage of the height or along the profile. Allowed values are in the range of 0% to 100%.

**Use negative side tilt (Checkbox):** The tool can be aligned on the defined contact point using a positive and a negative side tilt angle with the same absolute value. By default, the positive angle will be used. Place a check in this check box to use the negative side tilt angle.



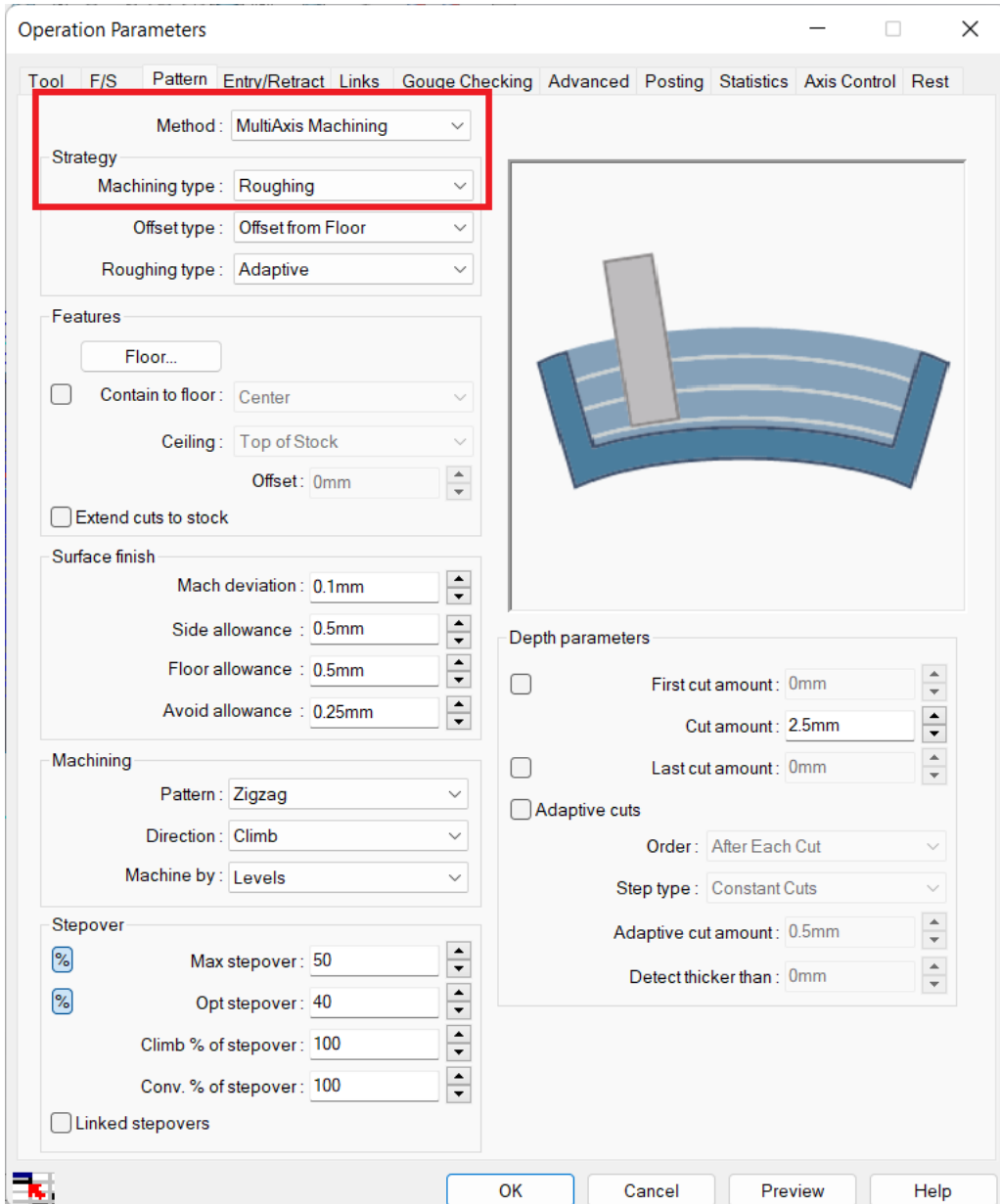
**New - Multiaxis Machining – Roughing Toolpath for Roughing out Pocket Geometries**

**Purpose:**

Option to generate a new type of Multiaxis Machining Roughing toolpath for roughing out pocket-shaped geometries

**Implementation:**

For a Multiaxis Mill operations, the type of Multiaxis toolpath to be generated can be assigned using the settings available under the **Pattern tab**. Under this tab, when the **Method** is set to **Multiaxis Machining** and the **Machining Type** is set to Roughing, a **Multiaxis Machining – Roughing** toolpath will be generated.



**Settings in Pattern tab of Multiaxis Operations to select Multiaxis Roughing**

The user must specify the floor surface(s) in order for CAMWorks to generate the roughing toolpaths.

This roughing cycle supports both Adaptive and Offset roughing types.

- **Adaptive**

Adaptive roughing strategy ensures that the cutting conditions remain almost constant. This offers great improvements compared to conventional constant offset Roughing strategies. This



strategy avoids full-width cuts by constantly measuring the engagement volume of the tool with material and by gradually removing material off the remaining stock. It guarantees a stable load on the tool, which allows an increased material removal rate at higher feedrates and reduces the overall machining time.

- **Offset**

Offset roughing applies a conventional constant offset roughing strategy.

### Supported Tool Types

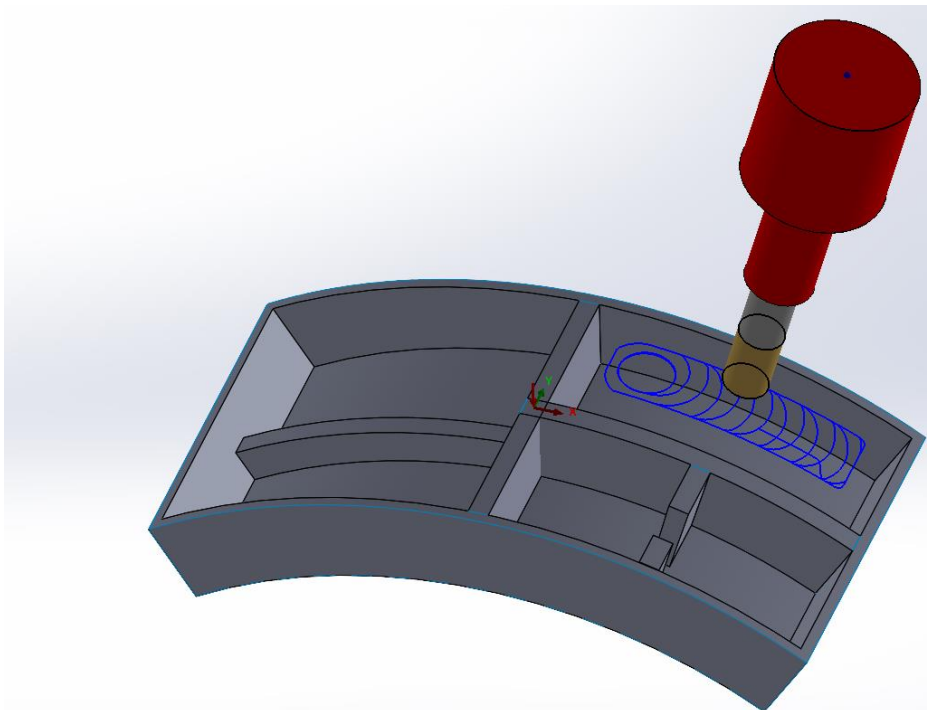
*Multiaxis Machining - Roughing* supports the following tool types:

- Flat End
- Ball Nose
- Hog Nose
- Flat End tapered
- Ball Nose tapered
- Hog Nose tapered

### Editing the settings for a Multiaxis Machining – Roughing Operation

The following tabs contain the parameters that are specific to *Multiaxis Machining - Roughing* operations and can be used to edit the settings for these type of operations:

- Pattern tab
- Entry/Retract tab
- Links tab
- Axis Control tab
- Gouge Checking tab
- Rest tab
- Advanced tab



**Sample Multiaxis Machining – Adaptive Roughing Toolpath for Machining a Pocket feature**



**New - Multiaxis Machining - Finishing Toolpaths for Finish Machining Pocket Geometries**

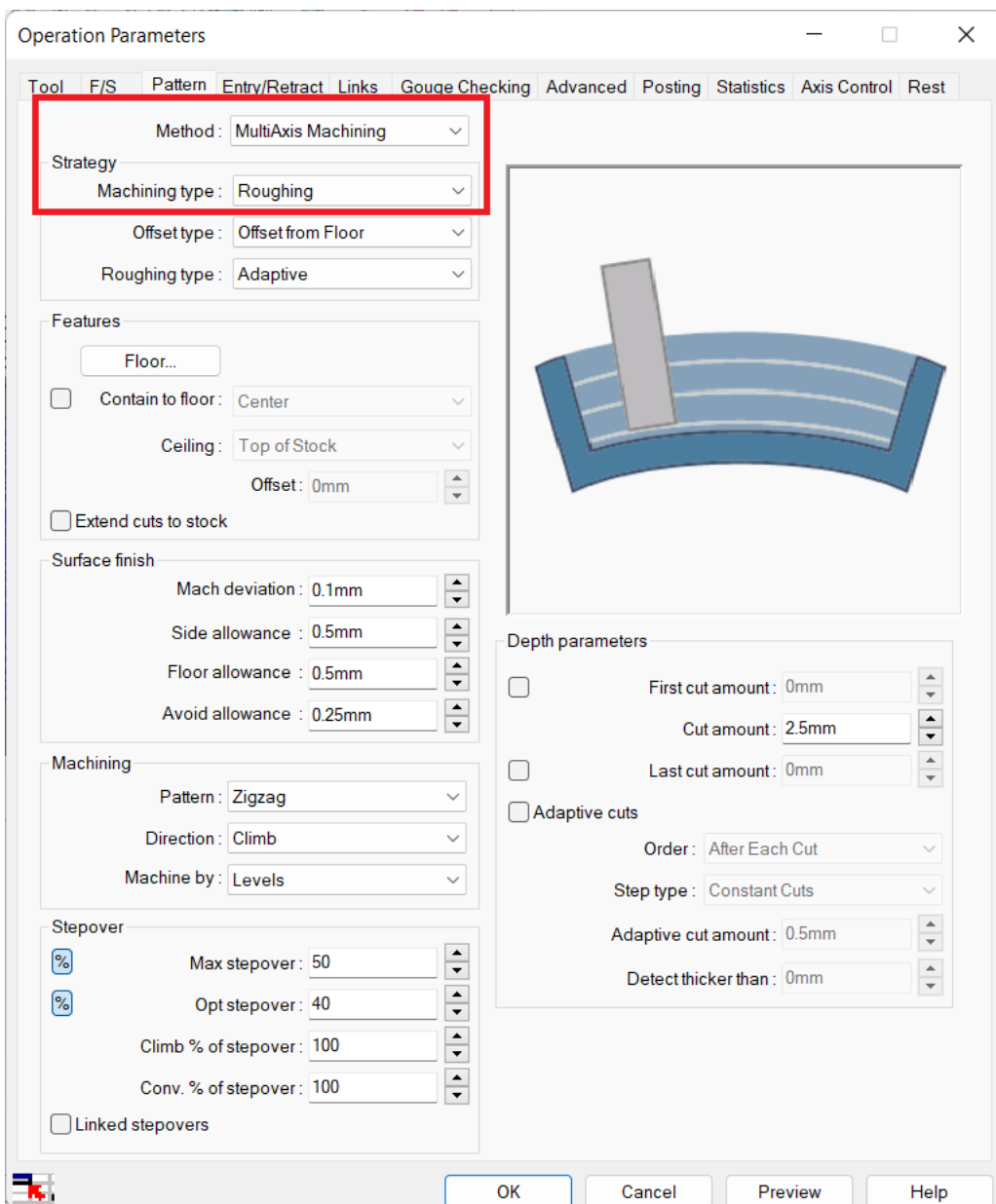
**Purpose:**

Option to generate a two new type of Multiaxis Machining – Finishing toolpaths for finish machining pocket-shaped geometries

**Implementation:**

For a Multiaxis Mill operations, the type of Multiaxis toolpath to be generated can be assigned using the settings available under the **Pattern tab**.

- Under this tab, when the **Method** is set to **Multiaxis Machining** and the **Machining Type** is set to **Floor Finishing**, a **Multiaxis Machining – Floor Finishing** toolpath will be generated. This toolpath will ideal for machining the floor of the pocket shaped geometries.
- Under this tab, when the **Method** is set to **Multiaxis Machining** and the **Machining Type** is set to **Wall Finishing**, a **Multiaxis Machining – Wall Finishing** toolpath will be generated. This toolpath will ideal for machining the floor of the pocket shaped geometries.



**Settings in Pattern tab of Multiaxis Operations to select Multiaxis Roughing**

The user must specify the floor surface(s) or wall surface(s) in order for CAMWorks to generate the corresponding finishing toolpaths.



### Supported Tool Types

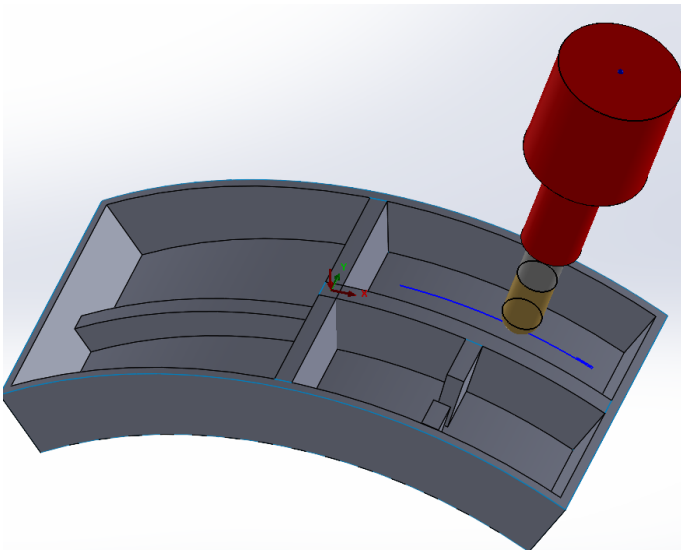
*Multiaxis Machining - Finishing* supports the following tool types:

- Ball Nose
- Ball Nose tapered
- Lollipop
- Barrel

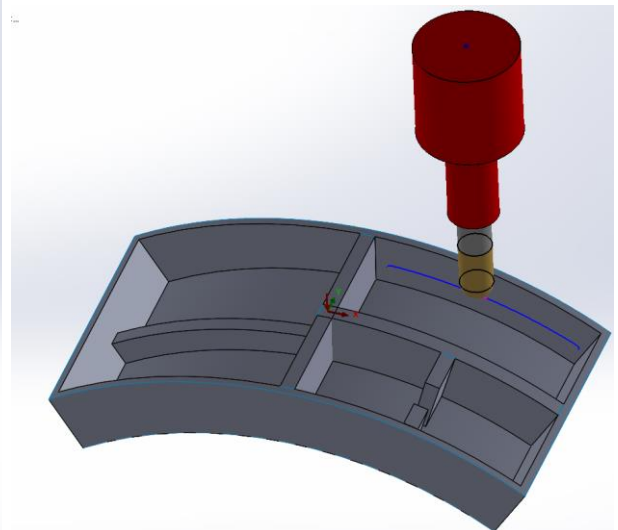
### Editing the settings for a Multiaxis Machining – Roughing Operation

The following tabs contain the parameters that are specific to *Multiaxis Machining - Roughing* operations and can be used to edit the settings for these type of operations:

- Pattern tab
- Entry/Retract tab
- Links tab
- Gouge Checking tab
- Advanced tab
- Axis Control tab



**Sample Multiaxis Machining – Floor Finishing  
Toolpath for Machining a Pocket feature**



**Sample Multiaxis Machining – Wall Finishing  
Toolpath for Machining a Pocket feature**





# Licensing

## **New - Message Prompt to upgrade License Manager on Floating Networks During CAMWorks Upgrade**

**Purpose:**

To prompt users to upgrade *CAMWorks License Manager* application on *License Server* machine of *Floating Network* installations to ensure successful license validation when client machines access licenses on the *License Server*

**Implementation:**

**Need to Upgrade CAMWorks License Manager Application**

The *CAMWorks License Manager* application is the tool that enables an installed *CAMWorks* application to verify and validate its configured license and associated licensed modules. Multiple minor upgrades and occasionally major upgrades are added to *CAMWorks License Manager* application supplied with every new *CAMWorks* release. If the upgrades are critical in nature, they will affect the license validation process. Hence, it becomes mandatory to upgrade the *CAMWorks License Manager* application along with the *CAMWorks* application to ensure successful license validation.

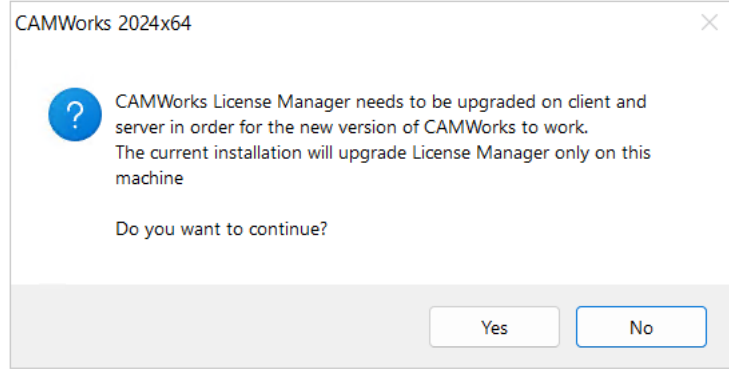
**License Validation Failure when CAMWorks License Manager Application is not Upgraded on License Server**

For Hardware dongle and node-locked licenses( both being single-user licenses), the *CAMWorks License Manager* application gets automatically updated when you install the upgraded *CAMWorks* version. However, for Floating Network licenses, the *CAMWorks License Manager* application needs to be upgraded on client machines as well as the *License Server*. If the *CAMWorks License Manager* version installed on the License Server is older than the version on a client machine, the license validation will fail and users of that specific client machine will not be able to use *CAMWorks*.

**Message Prompt to Ensure CAMWorks License Manager Application is Upgraded on License Servers**

From *CAMWorks 2024 BETA* version onwards, for *CAMWorks Floating Network* installations, a fail-safe in the form of a message prompt has been introduced to ensure that users upgrade the *CAMWorks License Manager* application on the client as well as the *License Server*. The fail-safe will work as follows:

1. On a client machine, when you run the executable file to install/upgrade the *CAMWorks* application, the *Installation Wizard* will first check whether the *CAMWorks License Manager* application is installed or not.
2. If the *CAMWorks License Manager* application is not installed, then it will proceed with the *CAMWorks* installation and also install the latest *CAMWorks License Manager* application.
3. If the *CAMWorks License Manager* application is installed, then the *Wizard* will check the application's version number.
  - i. If the version number of the installed *License Manager* application is compatible for running the *CAMWorks* version that will be installed by the executable file, then will proceed with the *CAMWorks* installation.
    - a. If a version of the *License Manager* application supplied with *CAMWorks* version being installed is newer to the version installed on the client, then the installed *License Manager* application will be upgraded.
    - b. If the version numbers are identical, then the *License Manager* application on the client will not be upgraded.
  - ii. If the version number of the installed *License Manager* application is not compatible for running the *CAMWorks* version that will be installed by the executable file, then the following message will be displayed:



**Notification Message Prompting Users to Upgrade CAMWorks License Manager**

- a. If you select **Yes** within this message box, then the *Installation Wizard* will proceed with *CAMWorks* installation and also upgrade the installed *License Manager* application. It is recommended that you upgrade the *License Manager* application installed on the *License Server*. Failure to do so will cause the license validation process between clients and License Server to fail.
- b. If you select **No** within this message box, then the *Installation Wizard* will exit. Upgrade the *License Manager* application on the *License Server* and then install the *CAMWorks* application on all the clients. When the message prompt is displayed again, select **Yes** to proceed with installation.