

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 © 2023-2024 HCL Technologies Ltd. All Rights Reserved.

October 2024



Table of Contents

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

What's New in CAMWorks 2024 SP5	4
Supported Platforms	4
Resolved CPRs Document	4
Compatibility of CAMWorks with SOLIDWORKS Apps Connected to 3DEXPERIENCE Platform	4
CAMWorks Utilities as a Partner Solution Add-in within SOLIDWORKS	5
What's New in CAMWorks 2024 SP4	6
Supported Platforms	6
Resolved CPRs Document	6
Compatibility of CAMWorks with SOLIDWORKS Apps Connected to 3DEXPERIENCE Platform	7
CAMWorks as a Partner Gold Add-in within SOLIDWORKS	8
What's New in CAMWorks 2024 SP3	9
Supported Platforms	9
Resolved CPRs Document	9
What's New in CAMWorks 2024 SP2	10
Supported Platforms	10
Resolved CPRs Document	10
What's New in CAMWorks 2024 SP1	11
Supported Platforms	11
Resolved CPRs Document	11
What's New in CAMWorks 2024 SP0	12
Supported Platforms	12
Resolved CPRs Document	12
Major Enhancements	13
VoluTurn: The new high performance Turn Roughing option	13
Machine Aware Programming and Visualization of CNC Machine within CAMWorks Environment	16
Multiaxis Machining – Roughing Toolpath for Roughing out Pocket Geometries	19
Multiaxis Machining - Finishing Toolpaths for Finish Machining Pocket Geometries	21
Dynamic Highlighting of Nodes in Feature Tree, Operation Tree and Tool Tree	24
Mill	27
Upper and Lower Driving Point for Bore, Dovetail, Keyway, and User- Defined Tools	27
Option to Suppress Top and Bottom Fillets of Mill Features for VoluMill Operations	29
'Avoid Part Faces Option for 2.5 Axis VoluMill Toolpaths	31
Option to define Cut Amounts as Percentage of Flute Length for 2.5 Axis Mill Operations	33
Improved Options for Condensed Hole Features	35
Optimized Pecking Option for Drill Toolpaths generated for Condensed Hole Features	38
Option to Generate Contour Mill Toolpaths That Machine The Features Form Bottom To Top	40
Turn/Mill-Turn	43
Highlight Recognized/Unrecognized Faces of the Turn Part	43
Defining Main and Sub Spindle Coordinate System (FCS) in Turn Assembly Mode Using SOLIDWORKS Coordinate Systems	44
Turn Tool Block Library In TechDB	45
Define Turn Operations on the Sub Spindle Setup at the start of the tree	48
Multiaxis	49
Tilt Tool by Contact Point option for Multiaxis Mill Operations	49
Tool Axis Smoothing Functionality for 5 Axis Mill Operations	51
Simulation	52
Dialog Box displaying Legend during Toolpath Simulation	52



TechDB	53
Option to assign default Virtual Simulation Machine to be associated with a Machine in TechDB	53



What's New in CAMWorks 2024 SP5

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: <ul style="list-style-type: none"> - SOLIDWORKS 2025 - SOLIDWORKS 2024 - SOLIDWORKS 2023 - CAMWorks Solids 2025 - CAMWorks Solids 2024 - CAMWorks Solids 2023 - 3DEXPERIENCE SOLIDWORKS R2024x
Operating System:	64-bit version of: <ul style="list-style-type: none"> - Windows 11 - Windows 10 <p style="text-align: right;">[*Home Editions are not supported]</p> <p>Note: CAMWorks 2024 is supported only on 64-bit Operating systems.</p>

Resolved CPRs Document

Purpose of Document:	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 (SP5).
Path to Document:	To view the document, select: <i>C:\Program Files\CAMWorks2024x64\CAMWorks_VC142\Lang\English\CW2024BuildInfo.pdf</i>

Compatibility of CAMWorks with SOLIDWORKS Apps Connected to 3DEXPERIENCE Platform

Partnership with SOLIDWORKS:	HCL, a member of the SOLIDWORKS Partner Program, has been officially authorized by Dassault Systèmes to develop, support, and promote add-ins for all SOLIDWORKS applications working with the cloud-based 3DEXPERIENCE platform (3DEXPERIENCE SOLIDWORKS, SOLIDWORKS with Cloud Services, and SOLIDWORKS 3D CAD with Collaborative Designer for SOLIDWORKS).
Implementation:	<p>CAMWorks is seamlessly compatible with SOLIDWORKS with Cloud Services, a bundle including the industry-leading SOLIDWORKS 3D CAD application and Collaborative Designer for SOLIDWORKS which connects it to the 3DEXPERIENCE platform, a unified cloud-based product development environment.</p> <p>CAMWorks works in the same way as SOLIDWORKS desktop application with one major enhancement: CAMWorks helps customers further harness the power of data management and cross-discipline collaboration in the cloud, streamlining your product development processes. CAMWorks 2024 SP4 and later versions can be launched and run as Add-ins within the 3DEXPERIENCE SOLIDWORKS application.</p>



CAMWorks Utilities as a Partner Solution Add-in within SOLIDWORKS

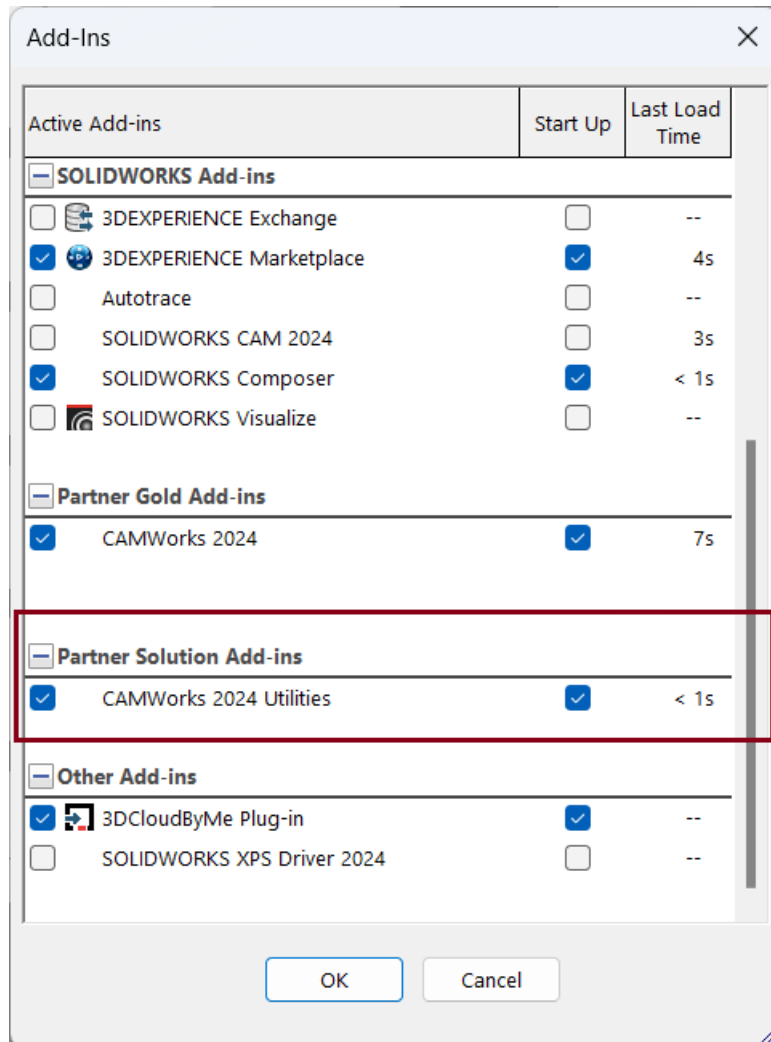
Purpose:

To ensure that the **CAMWorks Utilities** tool is listed under the **Partner Solution** Add-ins category within the *Add-ins* dialog box of the *SOLIDWORKS* application

Implementation:

The *CAMWorks Utilities* tool runs as an Add-In within the *SOLIDWORKS/ CAMWorks Solids* application. In previous versions of *CAMWorks*, the option to select *CAMWorks Utilities* as an add-in was available under the *Other Add-ins* category within the *Add-ins* dialog box of the *SOLIDWORKS* application.

CAMWorks 2024 SP5 version onwards, *CAMWorks Utilities* will be listed under the *Partner Solution Add-ins* category of *Add-ins* dialog box.



CAMWorks 2024 Utilities Add-in available under 'Partner Solution Add-ins' Category of Add-ins Dialog Box



What's New in CAMWorks 2024 SP4

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: <ul style="list-style-type: none"> - SOLIDWORKS 2024 - SOLIDWORKS 2023 - CAMWorks Solids 2024 - CAMWorks Solids 2023 - 3DEXPERIENCE SOLIDWORKS R2024x
Operating System:	64-bit version of: <ul style="list-style-type: none"> - Windows 11 - Windows 10 <p style="text-align: right;">[*Home Editions are not supported]</p> <p>Note: CAMWorks 2024 is supported only on 64-bit Operating systems.</p>

Resolved CPRs Document

Purpose of Document:	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 (SP4).
Path to Document:	To view the document, select: C:\Program Files\CAMWorks2024x64\CAMWorks_VC142\Lang\English\CW2024BuildInfo.pdf



Compatibility of CAMWorks with SOLIDWORKS Apps Connected to 3DEXPERIENCE Platform

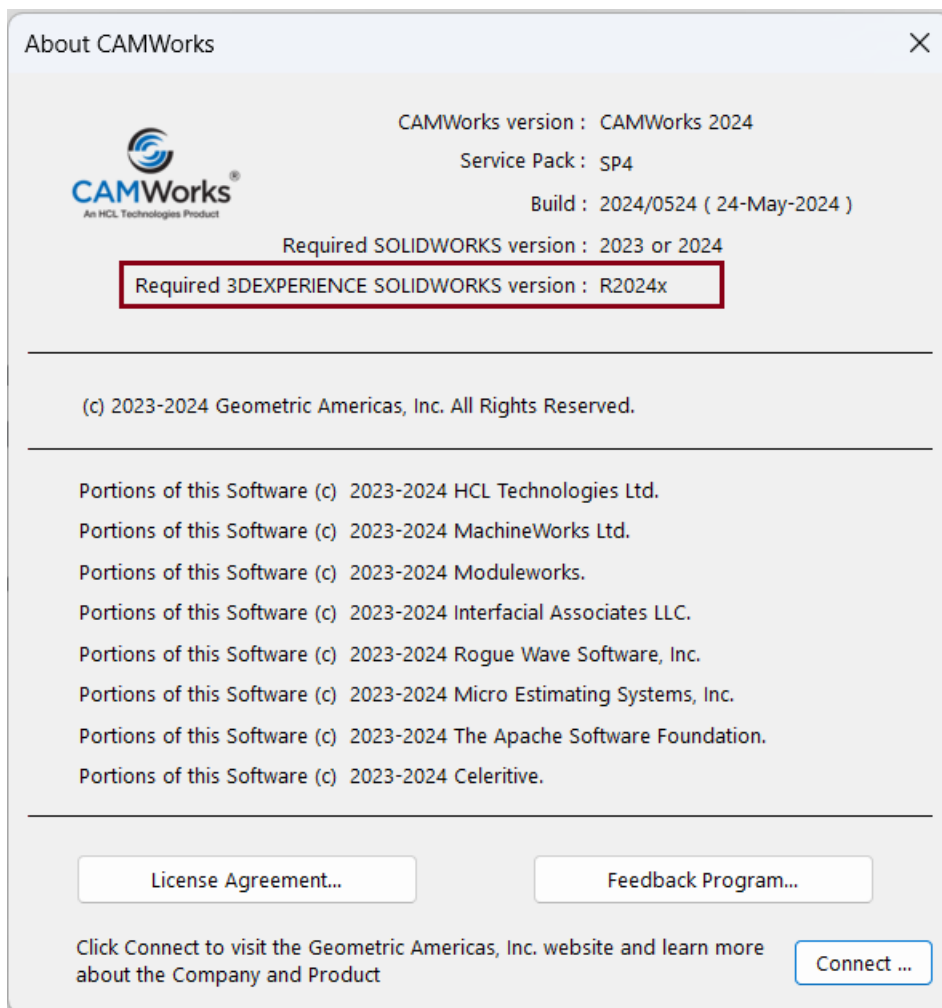
Partnership with SOLIDWORKS:

HCL, a member of the SOLIDWORKS Partner Program, has been officially authorized by Dassault Systèmes to develop, support, and promote add-ins for all SOLIDWORKS applications working with the cloud-based 3DEXPERIENCE platform (3DEXPERIENCE SOLIDWORKS, SOLIDWORKS with Cloud Services, and SOLIDWORKS 3D CAD with Collaborative Designer for SOLIDWORKS).

Implementation:

CAMWorks is seamlessly compatible with SOLIDWORKS with Cloud Services, a bundle including the industry-leading SOLIDWORKS 3D CAD application and Collaborative Designer for SOLIDWORKS which connects it to the 3DEXPERIENCE platform, a unified cloud-based product development environment.

CAMWorks works in the same way as SOLIDWORKS desktop application with one major enhancement: CAMWorks helps customers further harness the power of data management and cross-discipline collaboration in the cloud, streamlining your product development processes. CAMWorks 2024 SP4 and later versions can be launched and run as Add-ins within the 3DEXPERIENCE SOLIDWORKS application.



About CAMWorks dialog box with 3DEXPERIENCE SOLIDWORKS R2024x as a supported version



CAMWorks as a Partner Gold Add-in within SOLIDWORKS

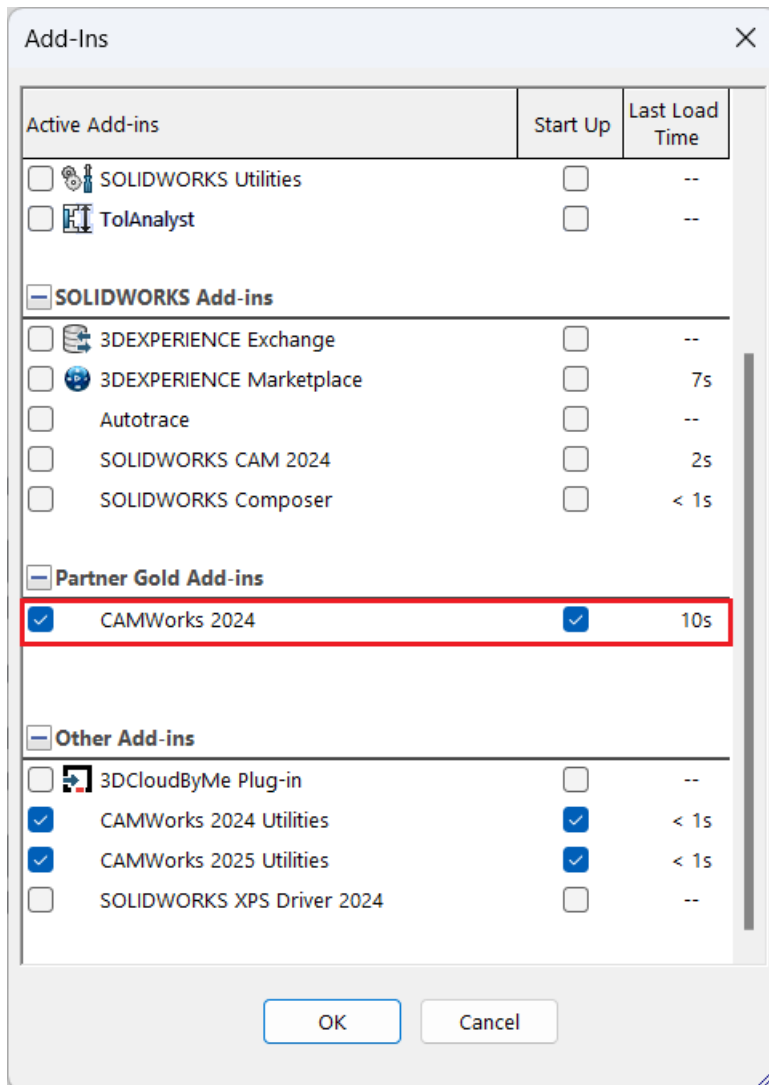
Purpose:

To ensure that the CAMWorks application is listed under the Partner Gold Add-ins category within the Add-ins dialog box of the SOLIDWORKS application

Implementation:

The CAMWorks application runs as an Add-In within the SOLIDWORKS/ CAMWorks Solids application. In previous versions of CAMWorks, the option to select CAMWorks as an add-in was available under the Other Add-ins category within the Add-Ins dialog box of the SOLIDWORKS application.

CAMWorks 2024 SP4 version onwards, CAMWorks will be listed under the Partner Gold Add-ins category of Add-Ins dialog box.



CAMWorks 2024 add-in available under Partner Gold Add-ins Category of Add-ins Dialog Box



What's New in CAMWorks 2024 SP3

Supported Platforms

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

Resolved CPRs Document

Purpose of Document:	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 (SP3).
Path to Document:	To view the document, select: <i>C:\Program Files\CAMWorks2024x64\CAMWorks_VC142\Lang\English\CW2024BuildInfo.pdf</i>



What's New in CAMWorks 2024 SP2

Supported Platforms

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

Resolved CPRs Document

Purpose of Document:	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 (SP2).
Path to Document:	To view the document, select: <i>C:\Program Files\CAMWorks2024x64\CAMWorks_VC142\Lang\English\CW2024BuildInfo.pdf</i>



What's New in CAMWorks 2024 SP1

Supported Platforms

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

Resolved CPRs Document

Purpose of Document:	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 (SP1).
Path to Document:	To view the document, select: <i>C:\Program Files\CAMWorks2024x64\CAMWorks_VC142\Lang\English\CW2024BuildInfo.pdf</i>



What's New in CAMWorks 2024 SP0

Supported Platforms

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

Resolved CPRs Document

Purpose of Document:	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 (SP0).
Path to Document:	To view the document, select: <i>C:\Program Files\CAMWorks2024x64\CAMWorks_VC142\Lang\English\CW2024BuildInfo.pdf</i>



Major Enhancements

New - VoluTurn: The new high performance Turn Roughing option

Purpose:

VoluTurn is a powerful new toolpath technology that can significantly improve the machining of any part that is turned using round inserts and offers manufacturers the ability to significantly increase productivity by reducing cycle times while also reducing wear on cutting tools.

Highlights:

- Easier on machine-tools – always smooth, flowing motion
- Increased tool life – eliminates over engagement and dwelling
- Predictable tool life – distributes wear evenly on inserts
- Reduced machining loads – material entry and exit is always smooth, circular, and tangential – no sharp corners
- Automatic collision avoidance, even with full holder assemblies
- Unidirectional or bidirectional cutting patterns, with constant or variable depths, fully utilize the inserts while helping prevent notch formation
- Efficient repositioning between cuts

Implementation:

VoluTurn™ is a high performance Turn Roughing toolpath option. It is an optional module within CAMWorks. Using the *VoluTurn* option for Roughing in place of traditional roughing methods will help reduce cycle times, extend tool life and reduce stress on machine tools.

To select *VoluTurn* as the roughing method of choice for a *Turn Roughing* toolpath, select the newly available *VoluTurn* option in the *Method* dropdown list under *Turn Rough* tab of the *Operation Parameters* dialog box.

Licensing Requirements for VoluTurn

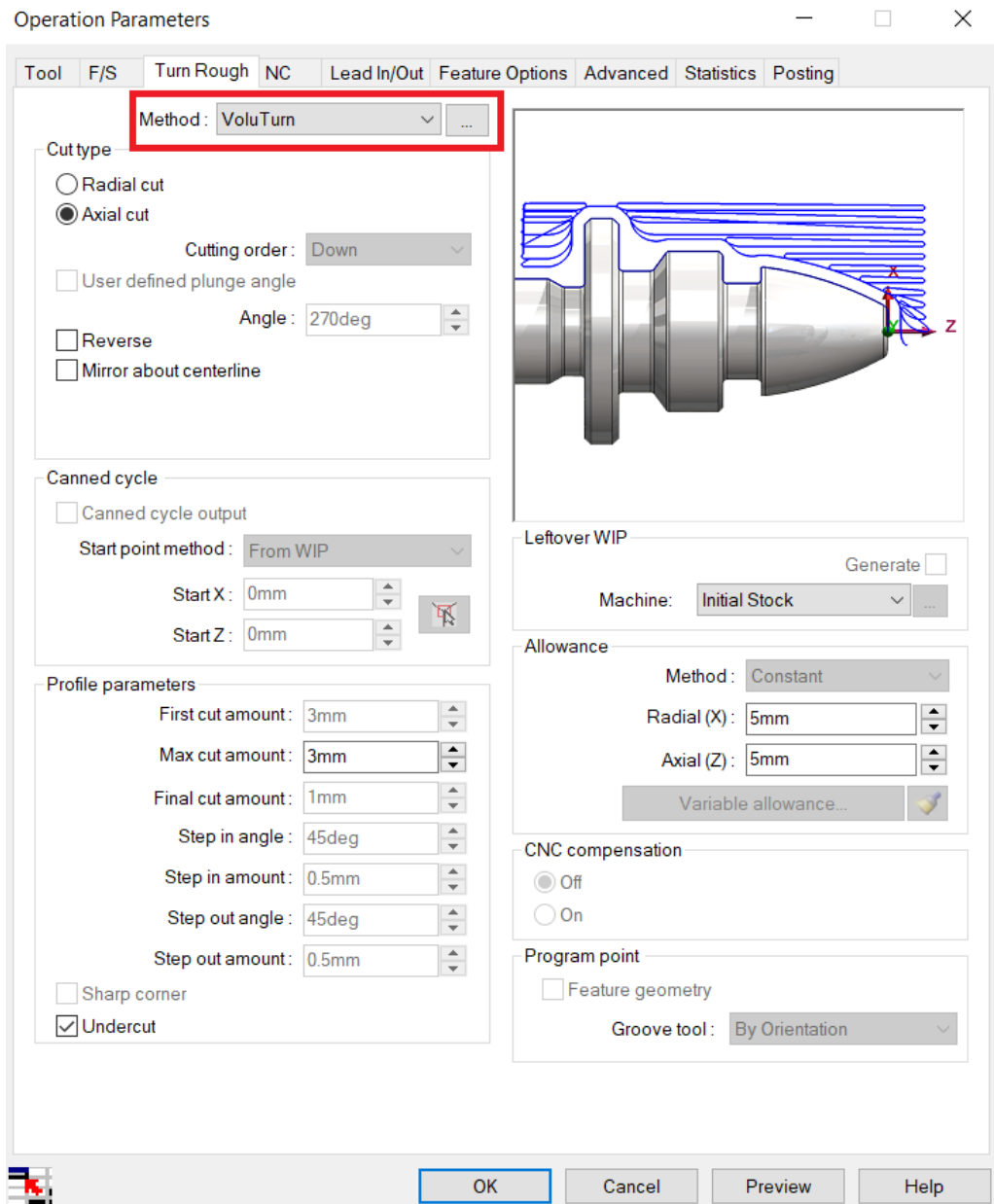
Your CAMWorks license needs to be configured to run the VoluTurn module. Without a valid license configured to run VoluTurn, you will not be able to generate, simulate and post process VoluTurn toolpaths. Contact your reseller to reconfigure your license.

VoluTurn Settings Dialog Box

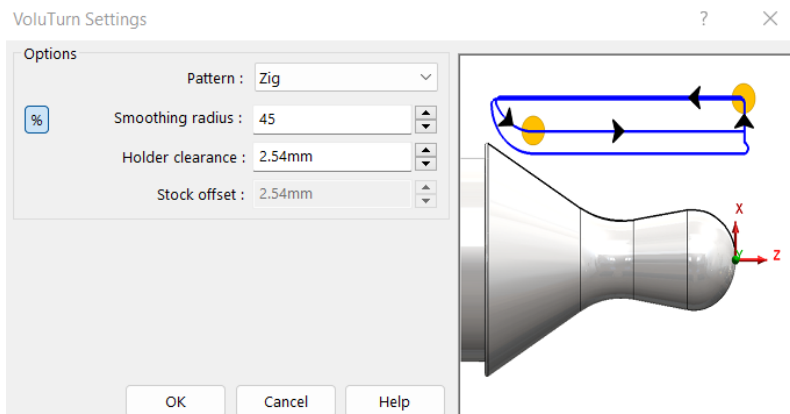
When *VoluTurn* option is selected in the *Method* dropdown list under the *Turn Rough* tab, the *Browse* button adjacent to the *Method* dropdown list will be enabled. Clicking on this button will display the *VoluTurn Settings* dialog box. Use this dialog box to edit settings that influence the VoluTurn toolpath. Use the **Pattern** parameter to control the pattern of the VoluTurn toolpath that will be generated. Use the **Smoothing radius** parameter to indicate the minimum radius the tool must transverse when cutting to machine into sharp corners or tight areas. The lowest value can be 5% of the insert diameter and max value can be equal to insert diameter. The default value will be 45% of the tool insert's diameter. The default value for this parameter can be modified in the **Turn Rough** Form within the TechDB.

Supported Tool Inserts:

VoluTurn option for roughing toolpaths is supported only for Turn Rough operations that are assigned Round turn inserts. If any other insert type is selected, then the VoluTurn toolpath will not be generated.



VoluTurn Option in Method Dropdown ist under Turn Rough tab in Operation Parameters Dialog Box



VoluTurn Settings Dialog Box



Operation Parameters

Save Copy Delete Move up Move down New Load Defaults

Operation

Id	Operation
1888	Turn Rough
1889	Turn Finish

Turn Rough

ID: 1888
Node description: ROUGH TURN

Feed/Speed

Turn Rough

Method: VoluTurn

Profile Parameters

Max cut amount: 3 mm
Undercut:

Allowance

Radial (X) allowance: 0.5 mm
Axial (Z) allowance: 0.5 mm

Leftover WIP

Generate:
From: Previous Leftover

Cut type

Cut type: Axial
Cutting order: Down

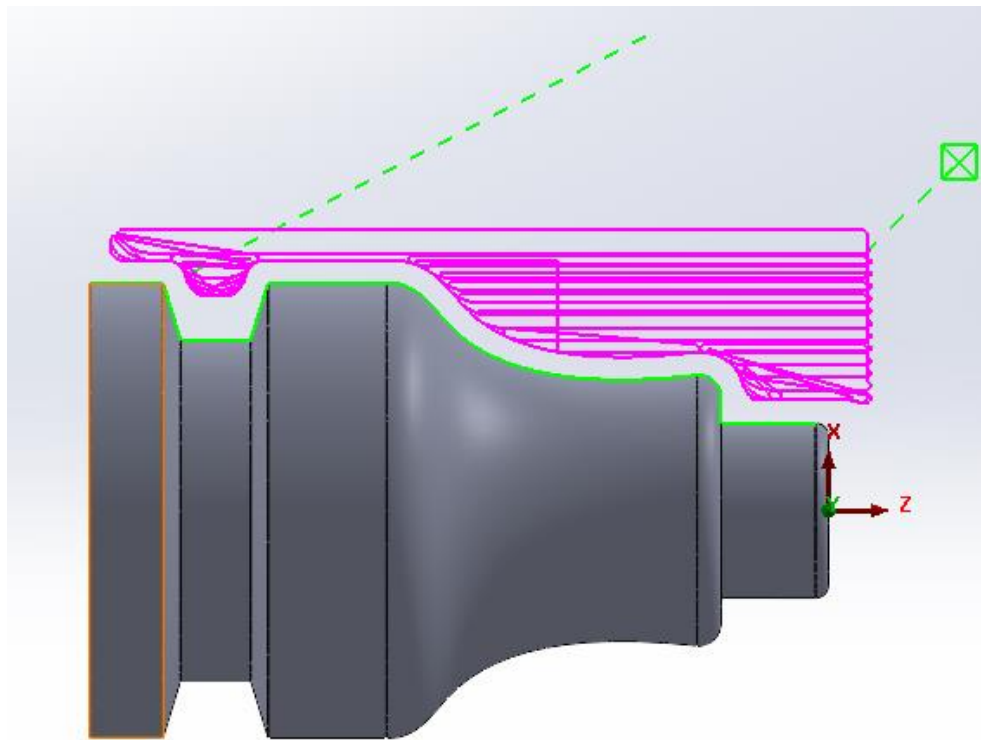
Reverse priority:
Mirror about centerline:
Simultaneous (Pinch) machining:

VoluTurn Settings

Options

Pattern: Zig
Smoothing radius: 45 %
Holder clearance: 2.54 mm
Stock offset: 2.54 mm

VoluTurn Settings within Turn Rough Form in TechDB



Sample VoluTurn Toolpath



New - Machine Aware Programming and Visualization of CNC Machine within CAMWorks Environment

Purpose:

Machine Aware NC Programming uses the exact digital twin of the CNC machine which will be used for machining the part. A NC programmer will be able to visualize the CNC machine withing the CAMWorks.

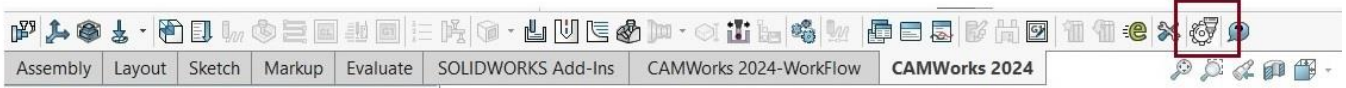
A NC programmer would setup the SOLIDWORKS part with respect to the virtual machine, the same way a machinist would setup a part on the physical machine.

NC programmer would be able to setup the virtual tools or the digital twins of the tools, needed for machining the part, the same way how a machinist would setup tools on the actual CNC machine.

Implementation:

Display Components toolbar:

A new command named **Display Components** will be available in the **CAMWorks Command Manager**. Executing this command will display the **Display Components** toolbar.



Display Components Command on CAMWorks Command Manager

The **Display Components** toolbar can be docked anywhere in the graphics area of the CAMWorks application. Use the following commands available within the **Display Components** toolbar to control the display of the various machine components comprising the Simulation Machine.

- **Display Machine (toggle button):** To control display of machine components
- **Display Housing (toggle button):** To control display of machine housing
- **Virtual Machine Stock Mounting (Command available only for Mill Machines):** Use this command to invoke the Define Point dialog box used for defining the virtual machine stock mounting point.
- **Display turret (toggle button available only for Turn/Mill-Turn machines):** To control display of the turret(s)
- **All tools (checkbox option):** To control the display of all the tools loaded on the turret(s) with option view translucent, shaded and shaded with edges display of the tools.

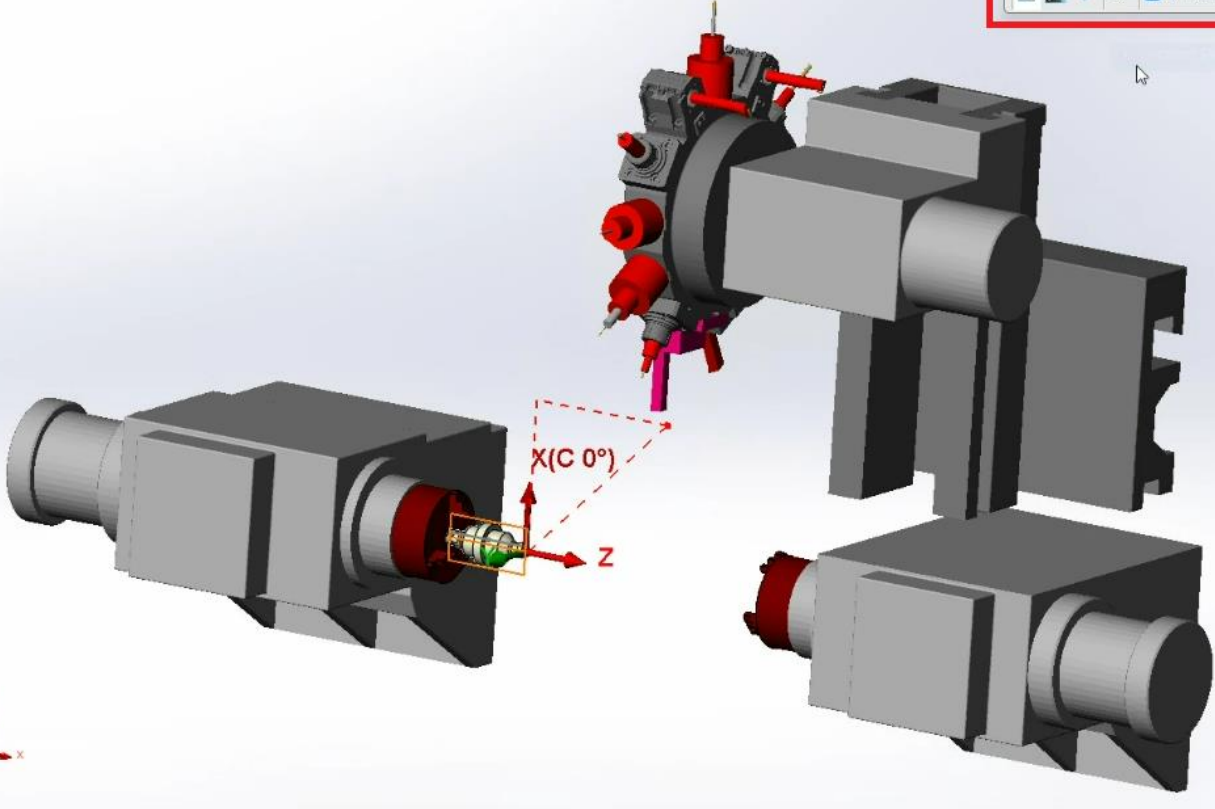
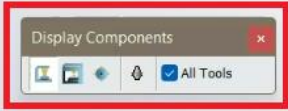
Machine Awareness and Visualization within the CAMWorks Environment

If you enable the **Machine display** option, the following will happen:

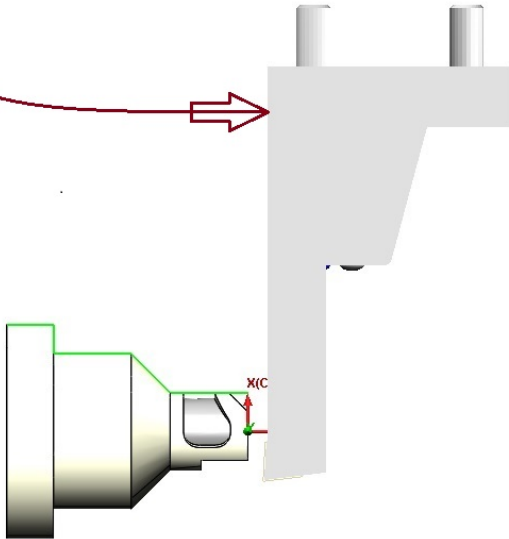
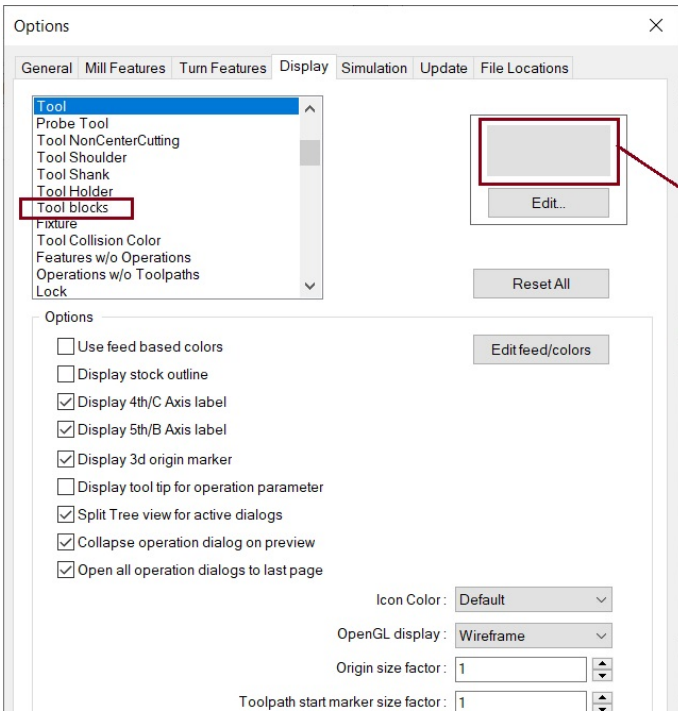
- When the **Holder** or **Station** sub-tab under the **Tool** tab is the active tab within the **Machine** dialog box or **Operation Parameters** dialog box, the tool, tool block and turret will be highlighted in the graphics area. Use the **Tool blocks** setting available under **Display** tab of the **CAMWorks Options** dialog box to assign color in which the tool block is to be displayed.
- When the **Tool Crib** sub-tab under the **Tool** tab is the active tab within the **Machine** dialog box or **Operation Parameters** dialog box, the tool heads and turrets will be highlighted in the graphics area.
- When the **Setup** or **Rotary** tab is the active tab within the **Machine** dialog box dialog box, the Tilt and Roarty axis, if defined, will be highlighted.
- When you mouse hover over any operation in the Operation tree, the tool assigned to that operation along with its tool block, and spindle will be highlighted in the graphics area.
- While editing or adding a sub spindle operation, if you elect any of the steps in the Sub Spindle Operation Parameters, then the action resulting from the selected step will be displayed in the graphics area along with machine components.

Note:

All the commands available in the **Display Components** toolbar will be disabled if no **Simulation Machine** is selected in the **Machine** tab of the **Machine** dialog box.



Display Components Toolbar with option to display Machine Components, Turret and Tools enabled

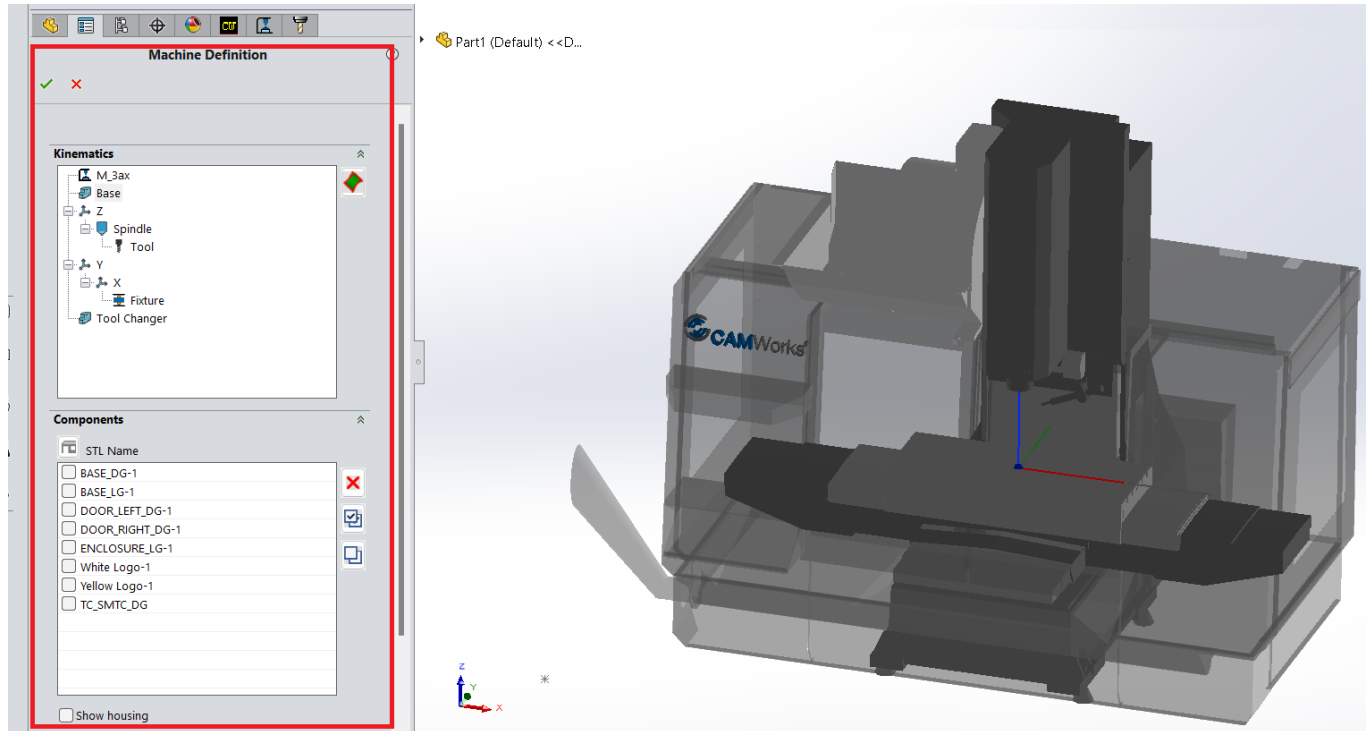


Setting to Assign Tool Bloxk Color under Display Tab of CAMWorks Options Dialog Box

Machine Definition Dialog box:

The **Machine Definition** dialog box is displayed when you click on the Import button under the Machine tab of the Machine dialog box.

Use the **Machine Definition** dialog box to control which components of the Simulation Machine are to be displayed in the graphics area and to categorize which of the Simulation Machine components as machine components or housing components.



Machine Definition Dialog Box



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

Purpose:

A new type of Multiaxis Machining Roughing toolpath for roughing out pocket-shaped geometries has been introduced in *CAMWorks*.

Roughing works the same as 3-axis roughing, the difference, or let us say the bonus is that the tool axis will be guided by the floor surface. This means that if the floor of a pocket is curved, the tool path will be five axes because the tool will always be vertical to the surface.

The algorithm supports end-mill, bullnose, and spherical tools that allow the machine of very complex parts with a very efficient roughing pattern. Users can choose between a pattern generation parallel to the ceiling or floor geometry or a morph pattern between both.

Key benefits:

- Quick and effortless way to define input parameters
- 5 Axis simultaneous cutting
- Collision checking against both part and in-process stock
- Stock-based rest roughing
- Undercut machining options

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.

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 stay 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

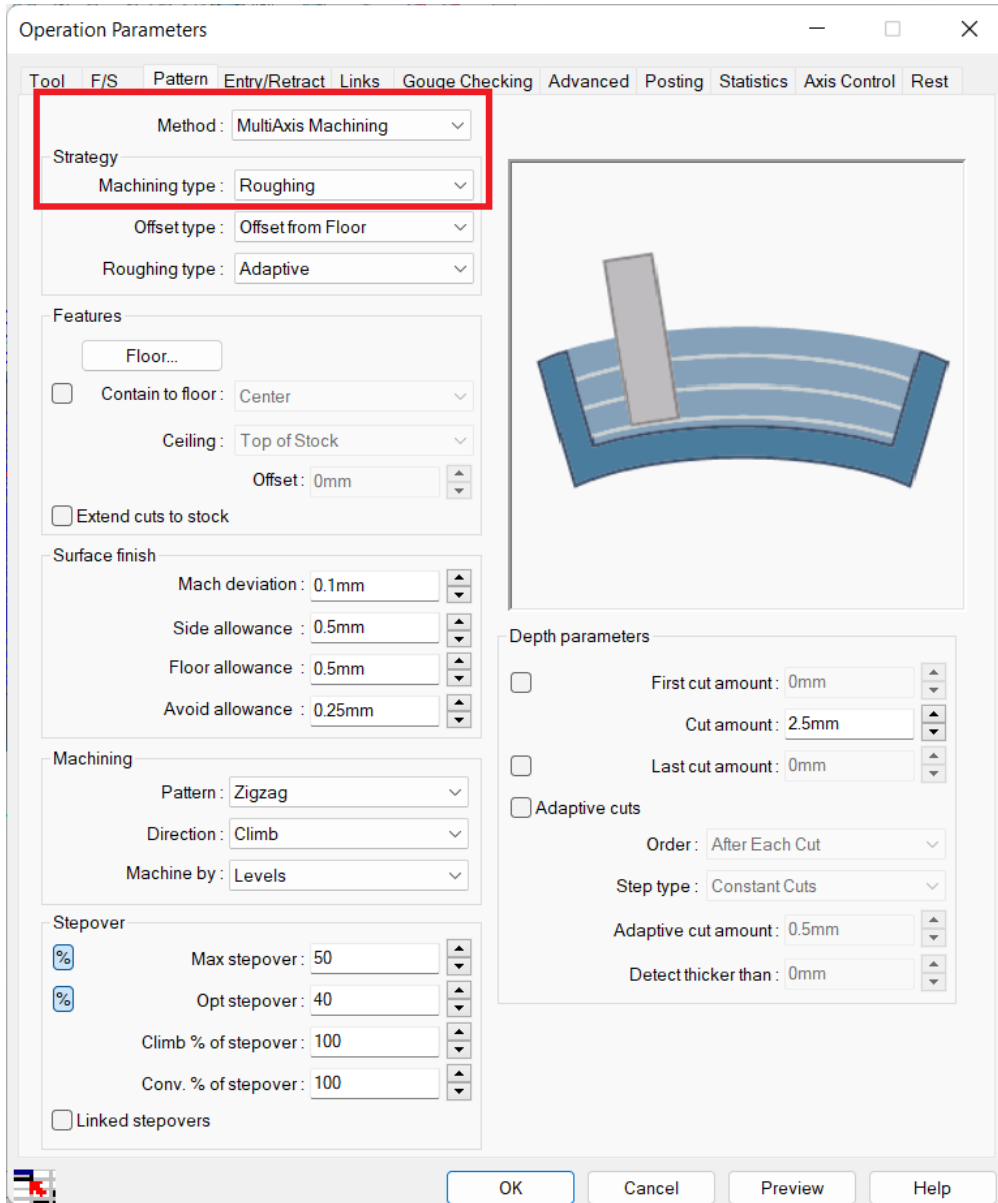
Multiaxis Machining - Roughing supports the following tool types:

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

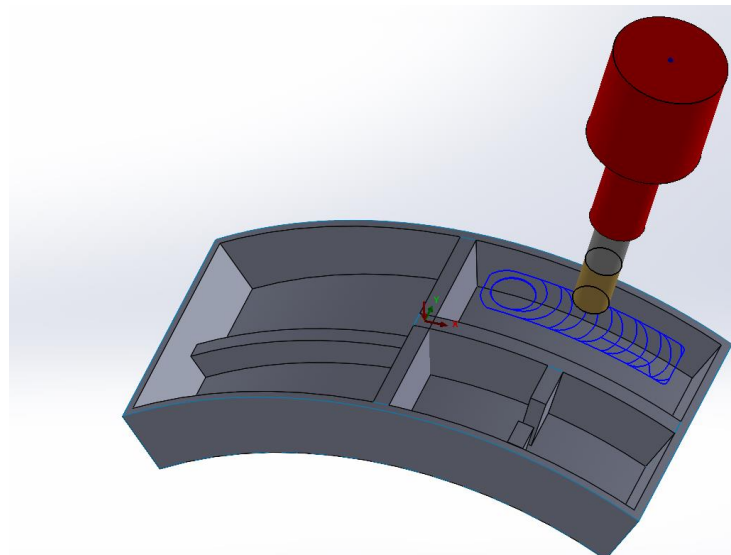
Editing the settings for a Multiaxis Machining – Roughing Operation

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

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



Settings in Pattern tab of Multiaxis Operations to select Multiaxis Roughing



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



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

Purpose:

A new type of *Multiaxis Machining – Finishing* toolpath for finish machining pocket-shaped geometries has been introduced in *CAMWorks*.

With Multi-axis Floor Finishing, *CAMWorks* provides a highly automated solution to create a floor-finishing toolpath for closed and open cavities. Due to the use of automatic tilting, the tool path is collision-free even for arbitrarily curved wall and floor surfaces. For barrel tools, the user can specify a desired tool contact point and an adjacent tool profile section that is allowed to be used to avoid collisions.

The algorithm supports spherical tools and lens barrel tools that allow a very efficient finishing with high step-over.

Key Benefits:

- High usability
- 5 Axis Simultaneous Machining
- Collision checking
- Optimized engagement control

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.

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

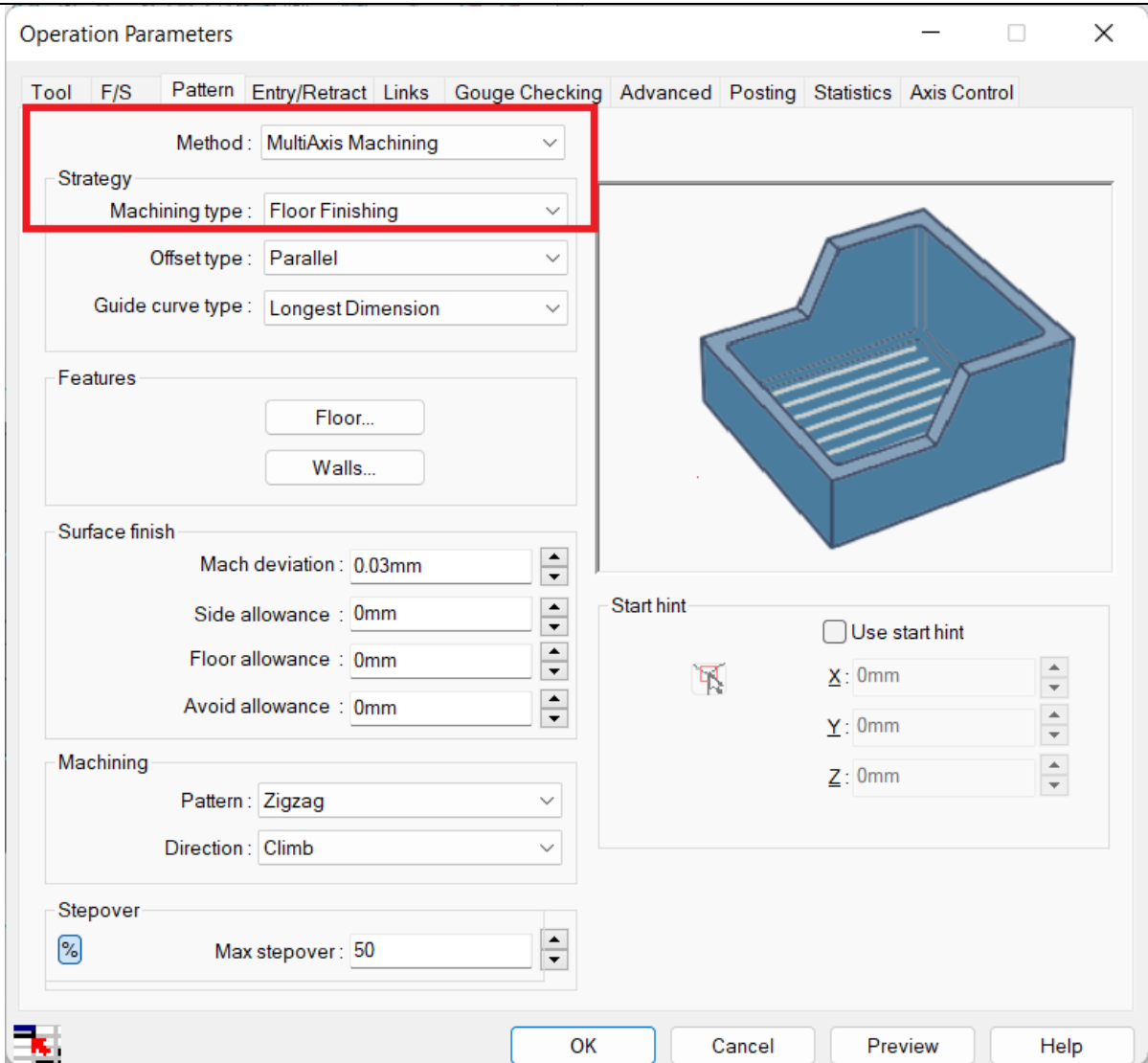
Multiaxis Machining - Finishing supports the following tool types:

- Ball Nose
- Ball Nose tapered
- Lollipop
- Barrel

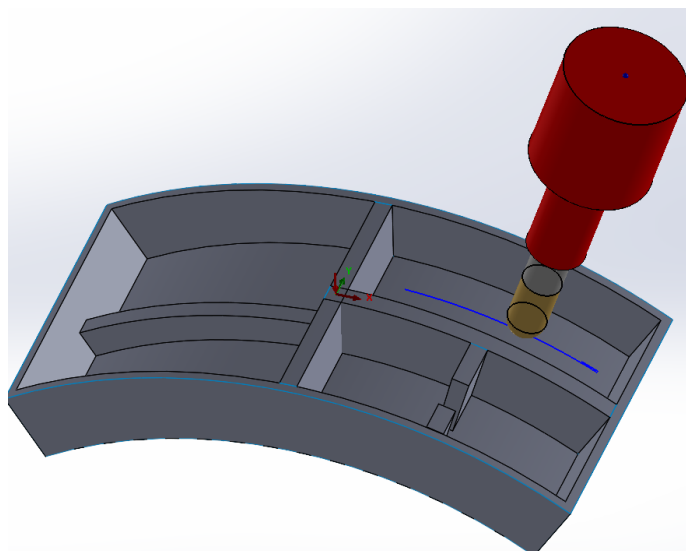
Editing the settings for a Multiaxis Machining – Finishing Operation

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

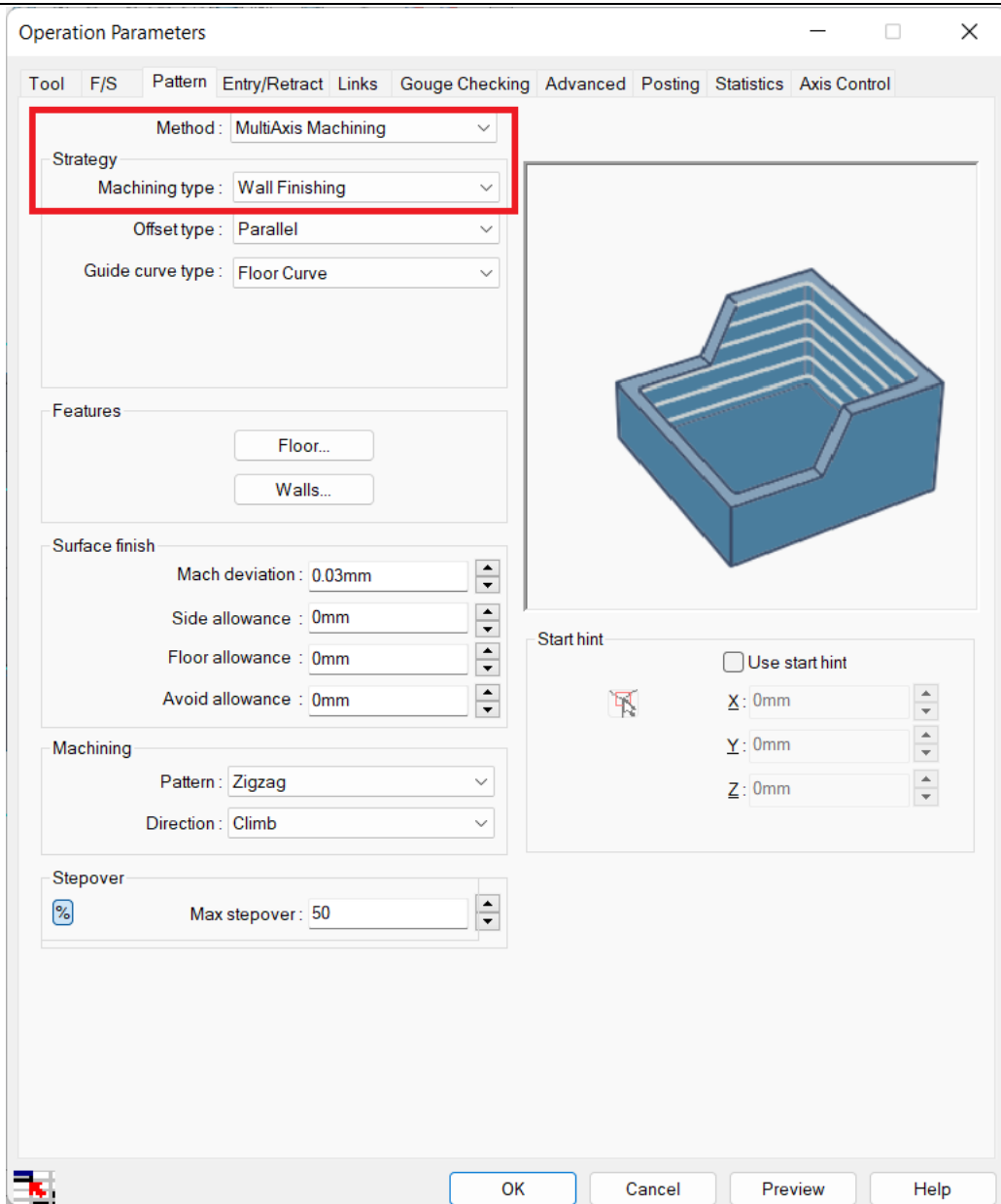
- Pattern tab
- Entry/Retract tab
- Links tab
- Gouge Checking tab
- Advanced tab
- Axis Control tab



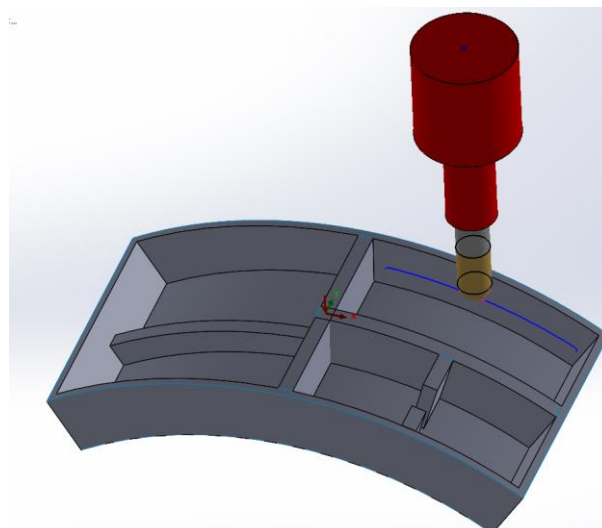
Settings in Pattern tab of Multiaxis Operations to select Multiaxis Floor Finishing



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



Settings in Pattern tab of Multiaxis Operations to select Multiaxis Wall Finishing



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



New - Dynamic Highlighting of Nodes in Feature Tree, Operation Tree and Tool Tree

Purpose:

CAMWorks has dynamic display of toolpaths or features when mouse hovers on the CAMWorks tree items.

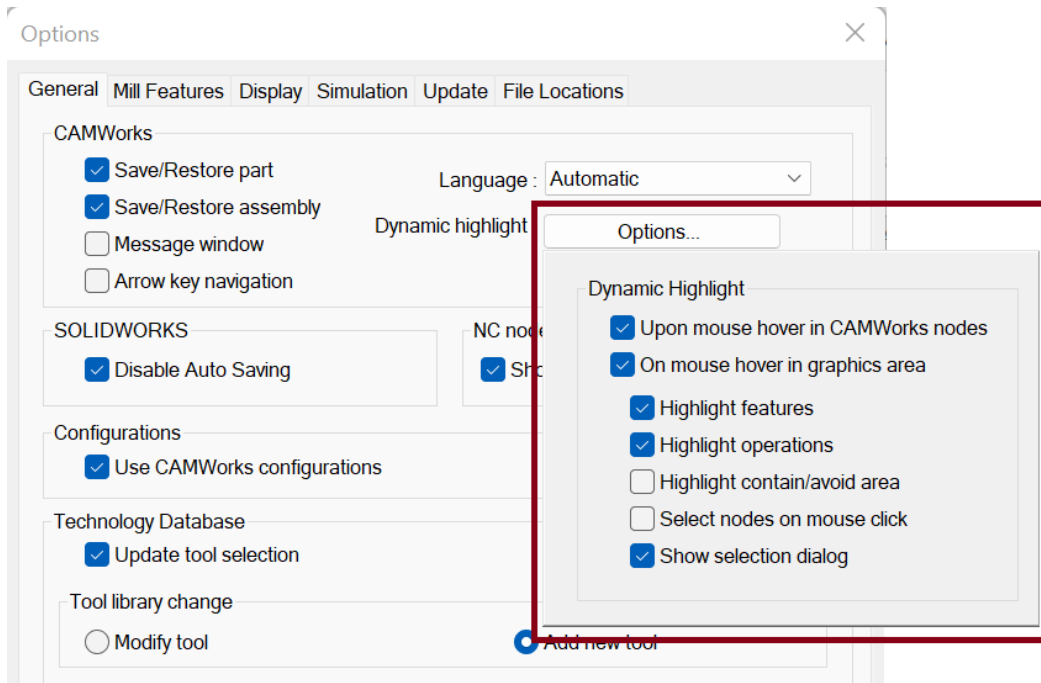
Now, it can be both ways, i.e., if mouse is on part faces, edges or sketches in the SOLIDWORKS graphics tree then it will highlight the associated CAMWorks tree items.

- In the Feature tree it can highlight the feature(s) when mouse is on the face or edge or a sketch.
- In the Operation tree, it can highlight the Operation(s) when mouse is on the face or edge or a sketch.
- In the Tools tree, it can highlight the Operation(s) when mouse is on the face or edge or a sketch.

Implementation:

In previous versions of CAMWorks, user could scroll through the CAMWorks tree and the corresponding entity would be highlighted in the graphics area. This was controlled using the Dynamic Highlight option in CAMWorks Options dialog box.

From CAMWorks 2024 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 CAMWorks nodes**

This checkbox option represents the existing functionality of dynamic highlighting.

• **On mouse hover in graphics area**

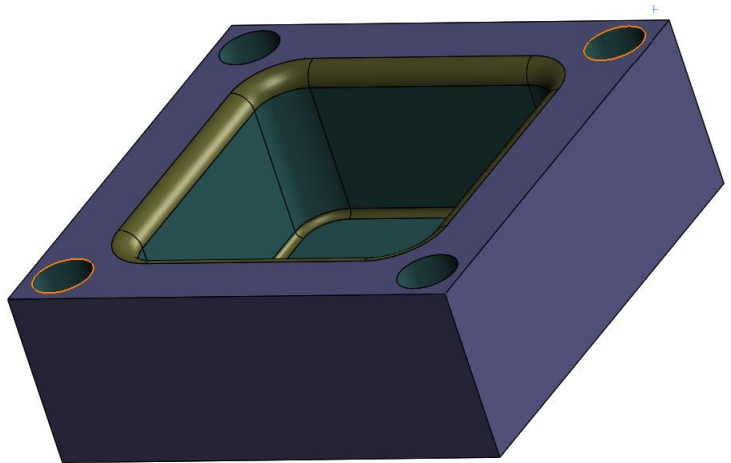
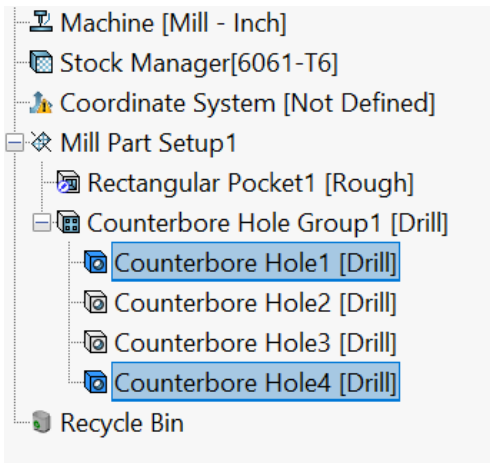
This checkbox has five sub-options associated with highlighting nodes in the Feature tree/ Operation tree when the entities associated with them are highlighted/selected on the part model in the graphics area. Selecting one or more of these sub-options will automatically place a check in this checkbox.



- **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 highlighted or selected in the graphics area. This selection can be in the 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 CAMWorks entities for the seed part and all other instances will be highlighted. 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 highlight the operation/s having contain or avoid areas defined using the highlighted or selected *SOLIDWORKS* entity in the display area.

- **Select nodes on mouse click**

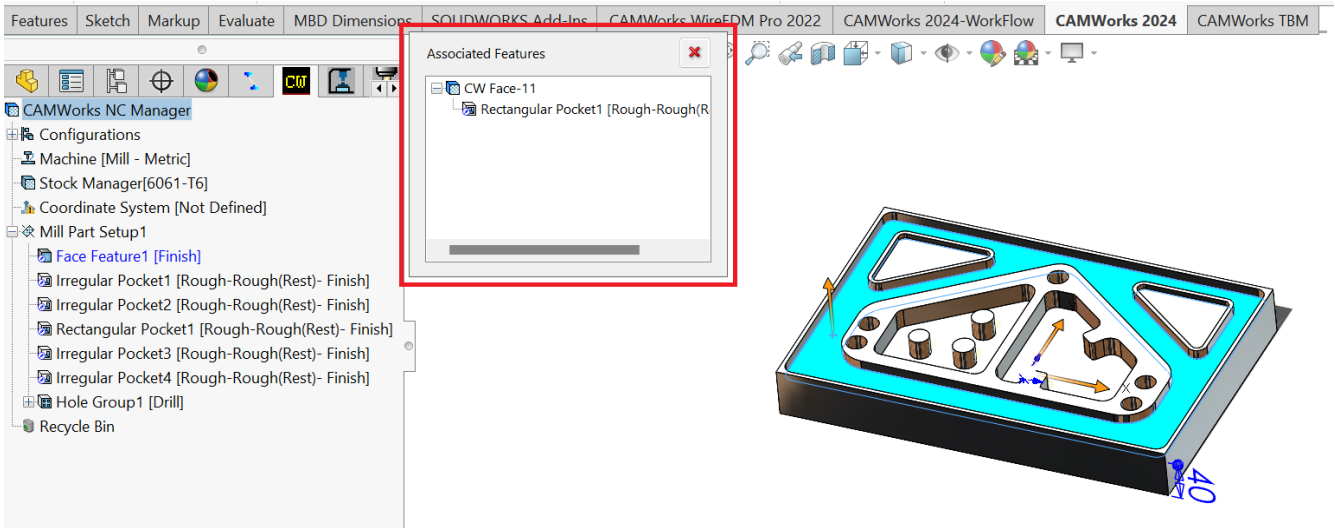
If user selects one or more faces on the part model or mouse hovers on any face of the part model/assembly in the graphics area, then the corresponding features or operations in the active tree will be selected and highlighted.

- **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

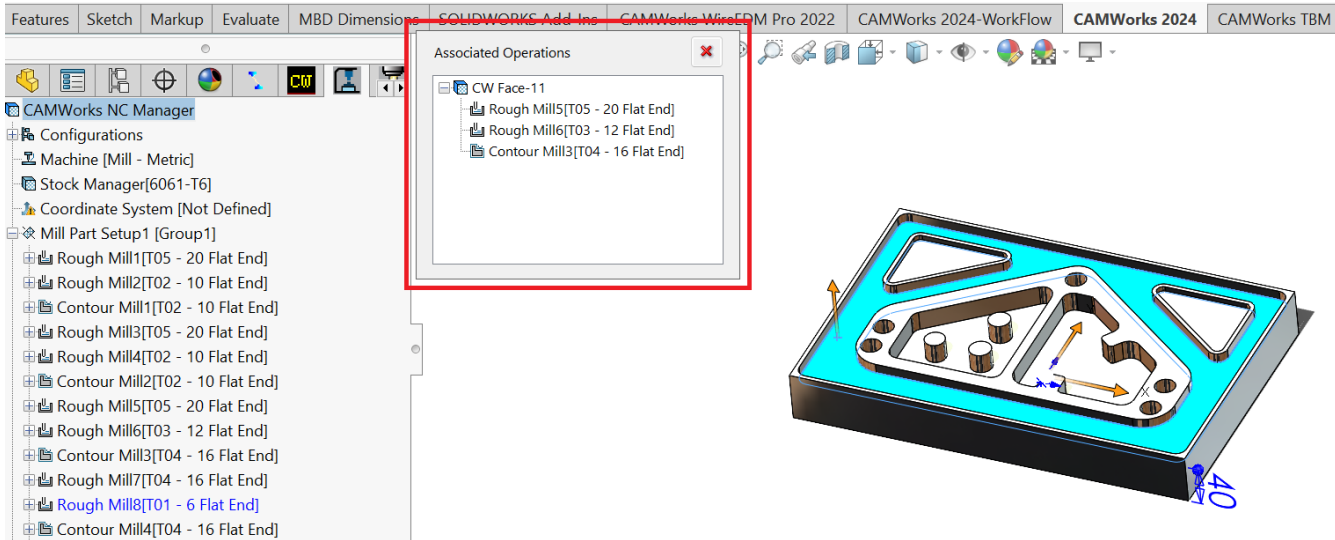
Within the *Associated Features* dialog box, the listed feature nodes will get dynamically updated as you mouse hover over other faces/edges or manually select other faces/edges of the part model/assembly. If you click on any feature node displayed within this dialog box, then its corresponding node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



Mill

Improved - Upper and Lower Driving Point for Bore, Dovetail, Keyway, and User-Defined Tools

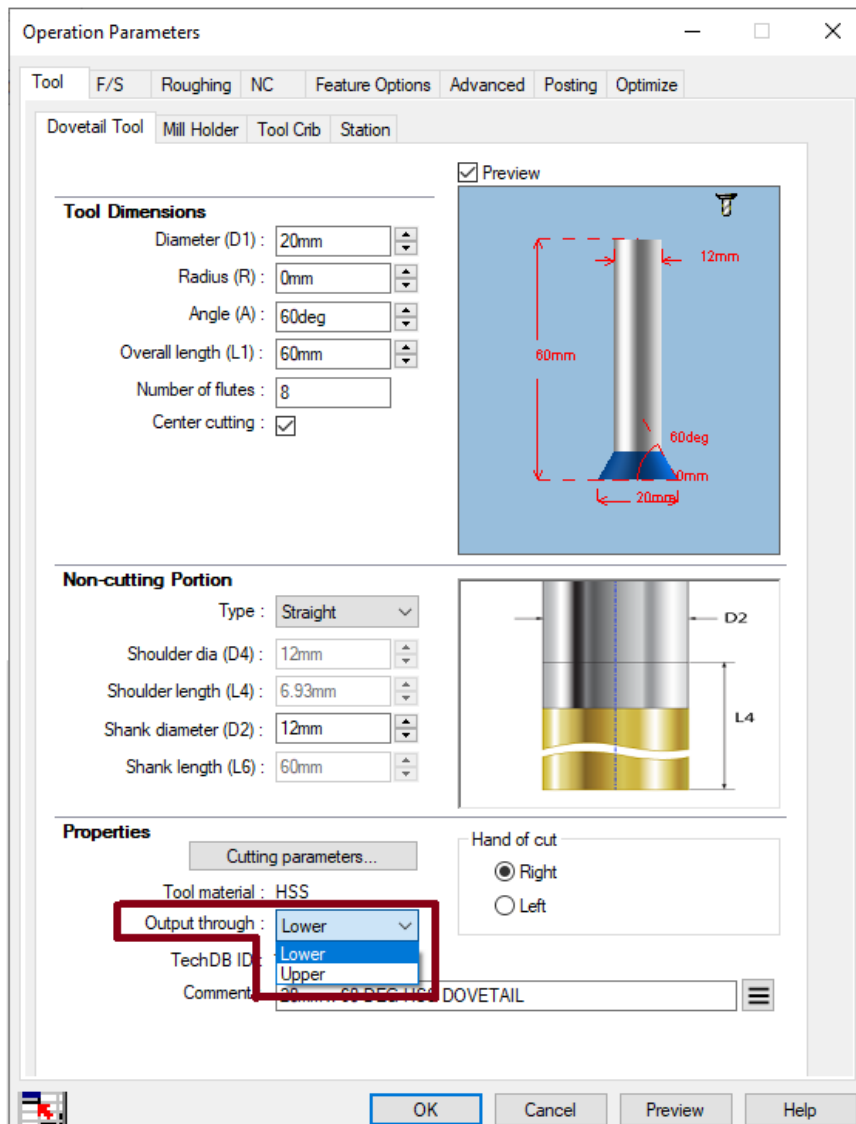
Purpose:

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

From CAMWorks 2024 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 generated using the above said tools.



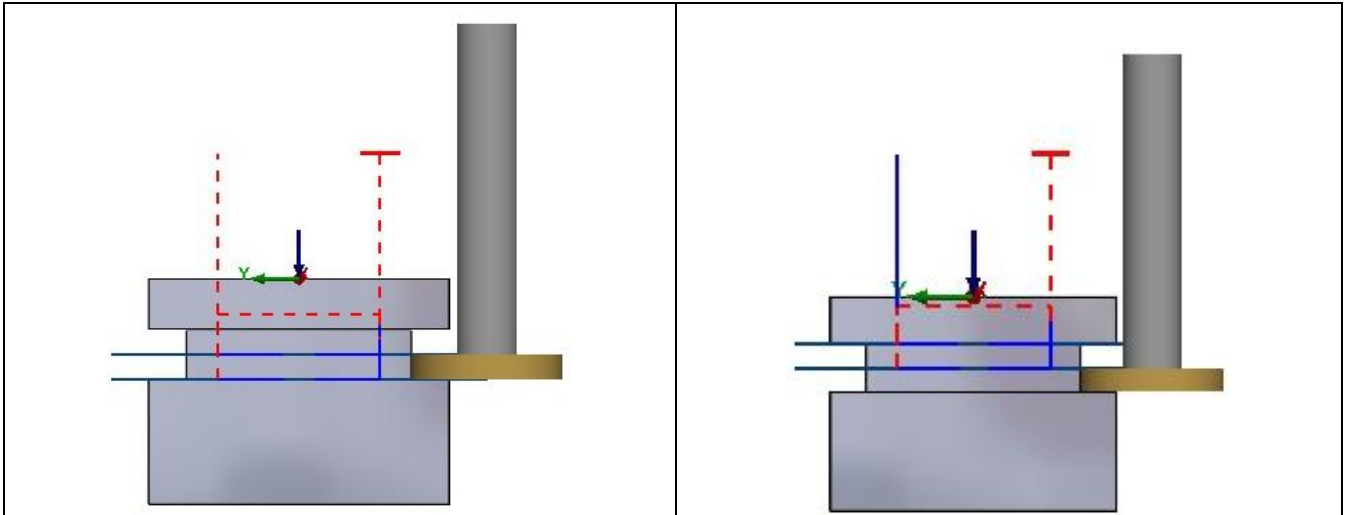
New 'Output Through' Option present under Tool Tab for Operations that Use Bore, Dovetail, Keyway or User-Defined Tools



Options in the 'Output Through' Dropdown List

Use this dropdown list option to specify the reference point for outputting G-Code coordinates of 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.



Illustrative Example of Keyway Tool when Output Through Option is set to 'Lower'

Illustrative Example of Keyway Tool when Output Through Option is set to 'Upper'

Note:

These output through options are not supported for Face Mill Operations.



New - Option to Suppress Top and Bottom Fillets of Mill Features for VoluMill Operations

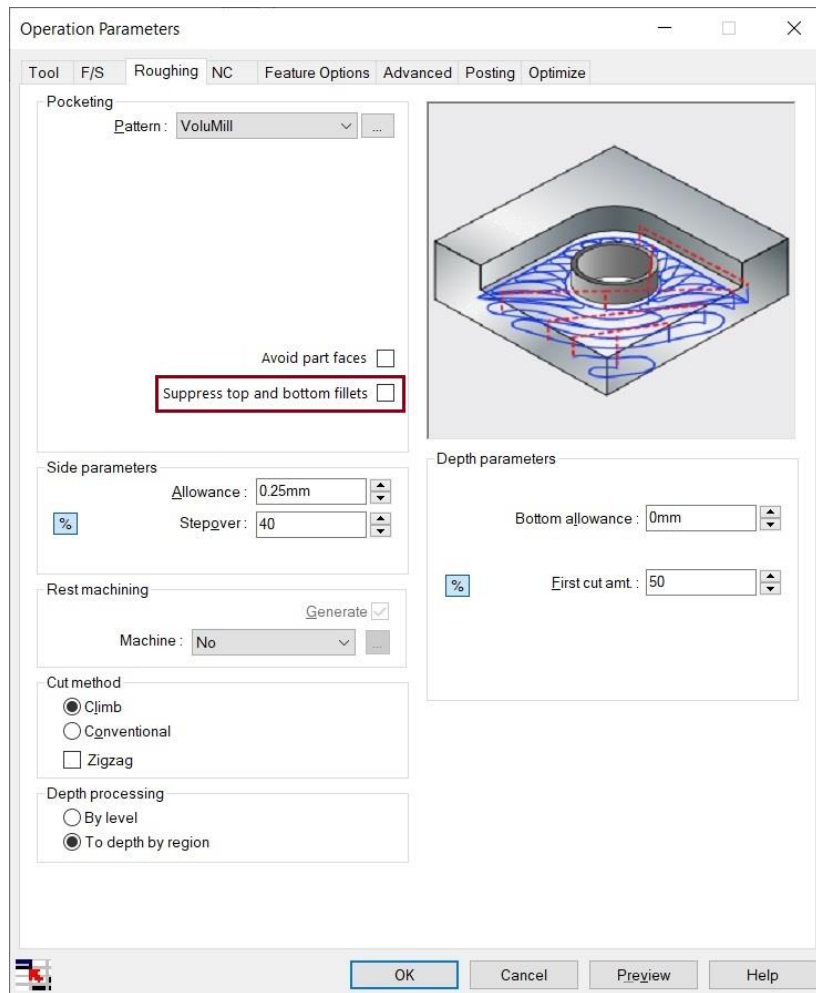
Purpose:

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 at the top and bottom, then the toolpath generated considered the geometry of the chamfers and/or fillets. There was no option to ignore the fillets and chamfers at the top and bottom of the feature.

From CAMWorks 2024 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** and only if the feature contains fillets at top and bottom edges. 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 SP0

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', including 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 (checked), and Machine island top (checked). The 'Suppress top and bottom fillets' checkbox is located at the bottom of the form and is highlighted with a red box.

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

Note:

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



New - 'Avoid Part Faces Option for 2.5 Axis VoluMill Toolpaths

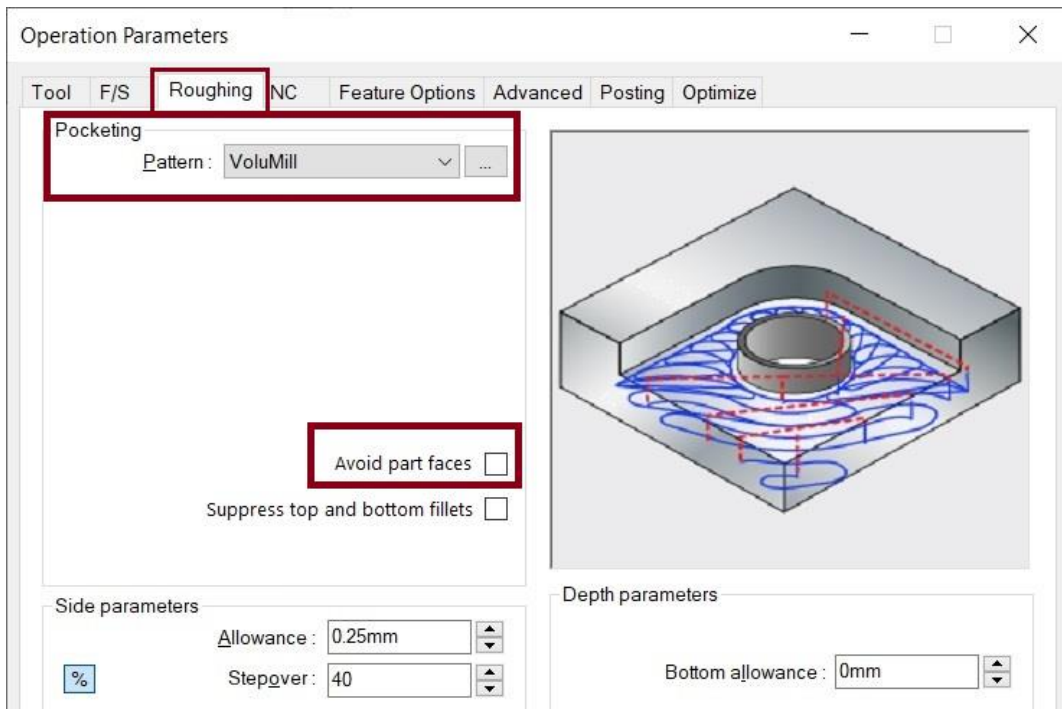
Purpose:

To ensure that in VoluMill toolpath the tool machining a given feature does not gouge the other faces which are not part of the feature being machined

Implementation:

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

From CAMWorks 2024 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 other faces of the part model as avoid faces. 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.

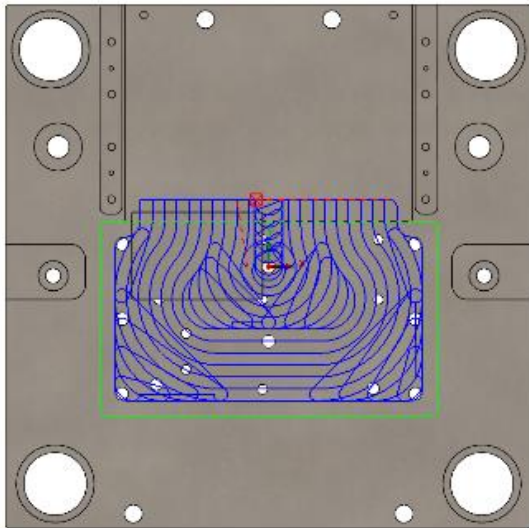
How 'Avoid part faces' Option works for VoluMill toolpath generated for an Assembly

If the VoluMill toolpath is generated for Mill features on any component part of an Assembly, then only eligible faces of that part instance 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.

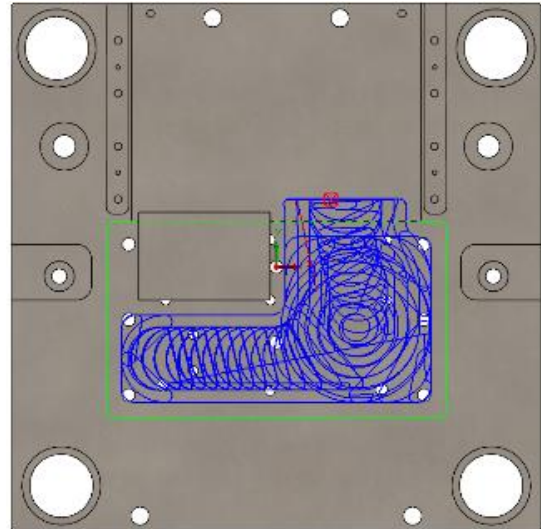


How 'Avoid part faces' Option works when one or more Avoided Faces are Suppressed

If any of the faces of the part are suppressed then, they are not considered as avoid faces by VoluMill even when the "Avoid part faces" check box is set to True.



**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:

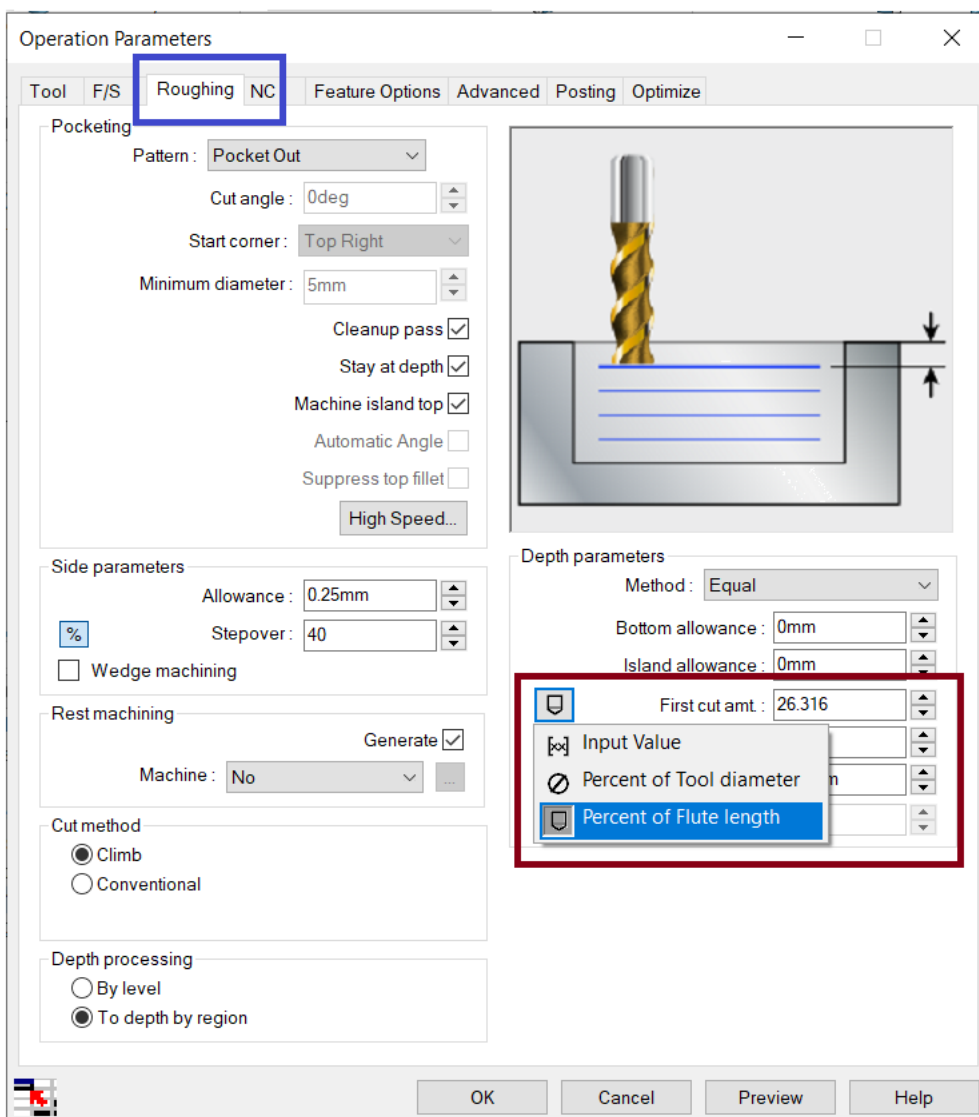
To provide an option that allows users to define *Cut Amount* parameters associated with 2.5 Axis Mill operations to be defined as a percentage of their assigned tool's flute length

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* 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






What's New in CAMWorks 2024 SP0

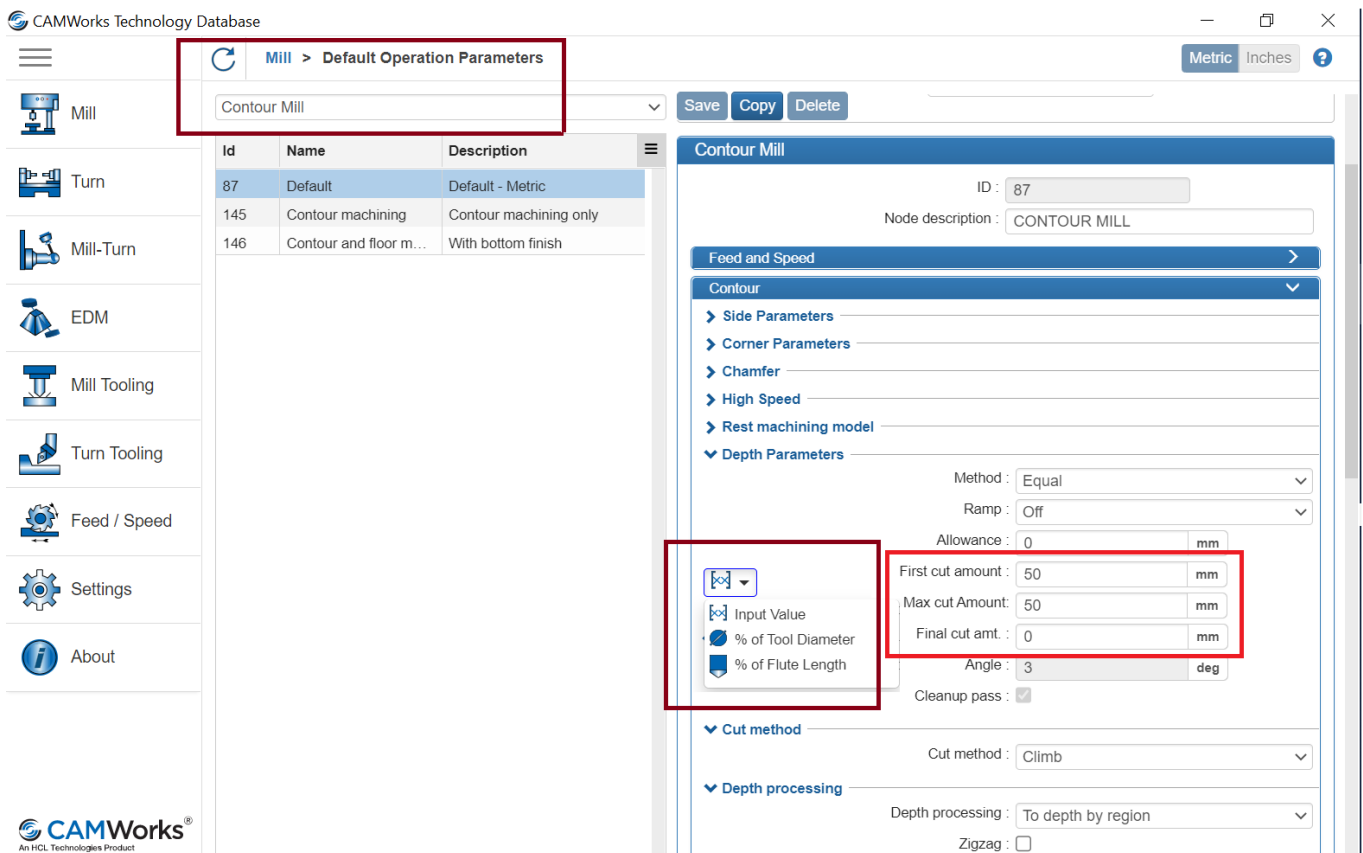
Within the above-listed user interfaces, the previously present **Percentage** toggle button has been replaced with a flyout button. Clicking on this button displays three options. Select the desired option for setting the method for assigning the **Cut amount** value.

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 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 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.



The screenshot displays the CAMWorks Technology Database interface. The main window is titled "CAMWorks Technology Database" and shows the "Default Operation Parameters" form for a "Contour Mill" operation. The interface includes a sidebar with navigation icons for Mill, Turn, Mill-Turn, EDM, Mill Tooling, Turn Tooling, Feed / Speed, Settings, and About. The main content area is divided into a table of parameters and a detailed parameter configuration panel. The table lists parameters with columns for Id, Name, and Description. The configuration panel is titled "Contour Mill" and includes fields for ID (87), Node description (CONTOUR MILL), and various parameters under "Feed and Speed" and "Depth Parameters". The "Depth Parameters" section is expanded, showing "Method: Equal", "Ramp: Off", "Allowance: 0 mm", and "First cut amount: 50 mm", "Max cut Amount: 50 mm", "Final cut amt.: 0 mm". A flyout menu is open, showing three options: "Input Value" (selected), "% of Tool Diameter", and "% of Flute Length".

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



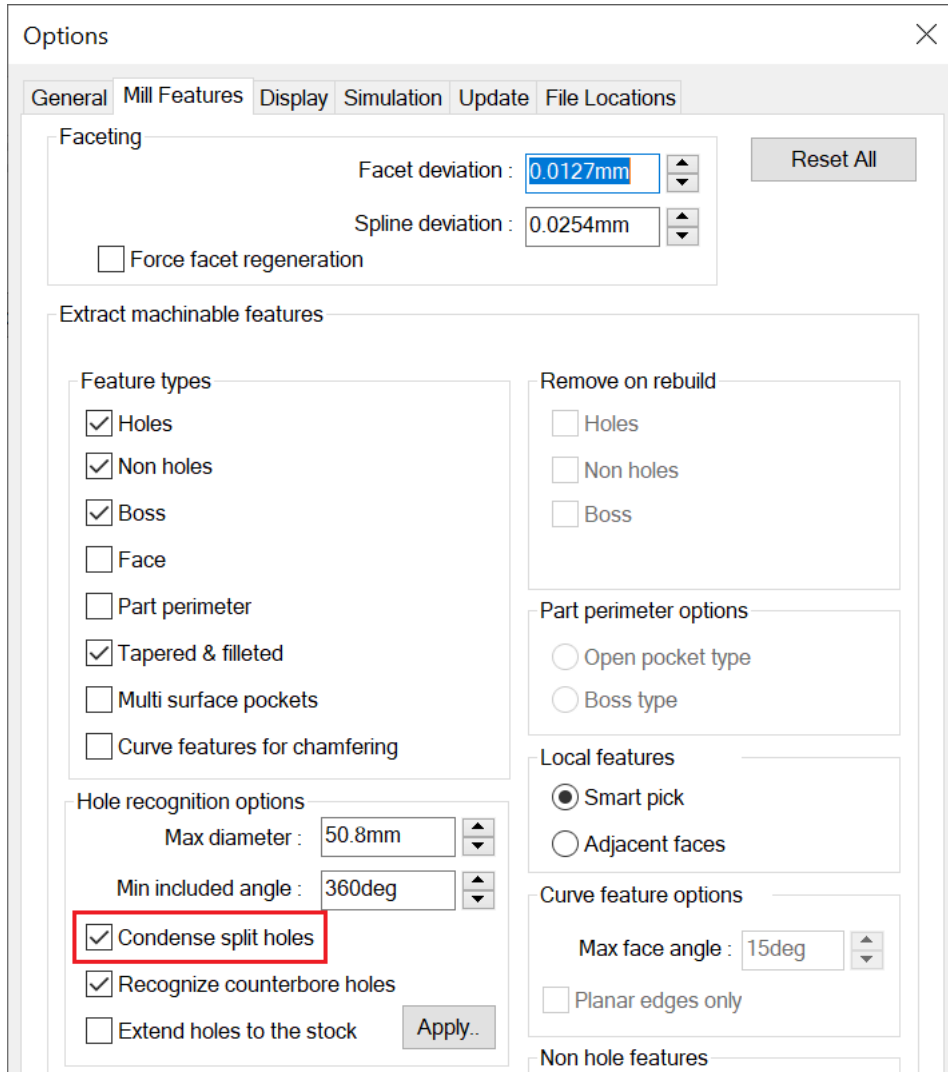
Improved - Improved Options for Condensed Hole Features

Purpose:

To provide the ability to interactively split/condense hole features. Also, display the number of holes/cylinders comprising condensed hole features and the parameters associated with the constituent hole features

Implementation:

In the current version of CAMWorks, user can condense the coaxial hole features using the option of **Condense split holes**. This option is available under the **Mill Features** tab of the **CAMWorks Options** dialog box.



Mill Features tab under CAMWorks Options Dialog Box

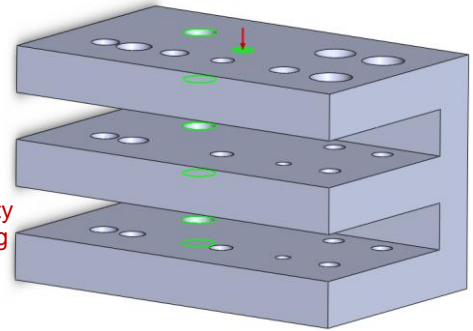
When this checkbox option is checked and *Automatic Feature Recognition* is executed, hole features which are co-axial and of same diameter are condensed into a single hole feature. This condensed hole will be represented by a single hole feature node in the Feature tree.

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

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



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

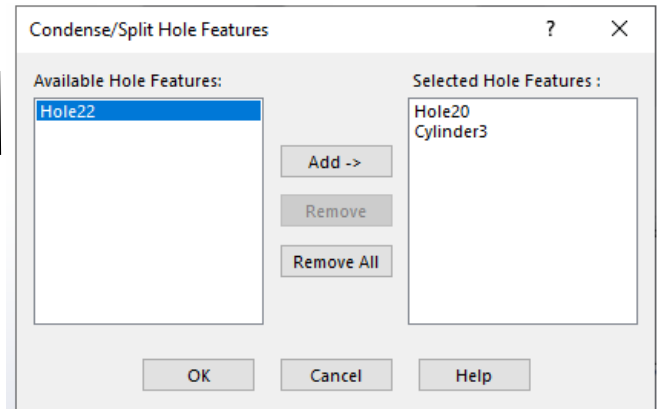
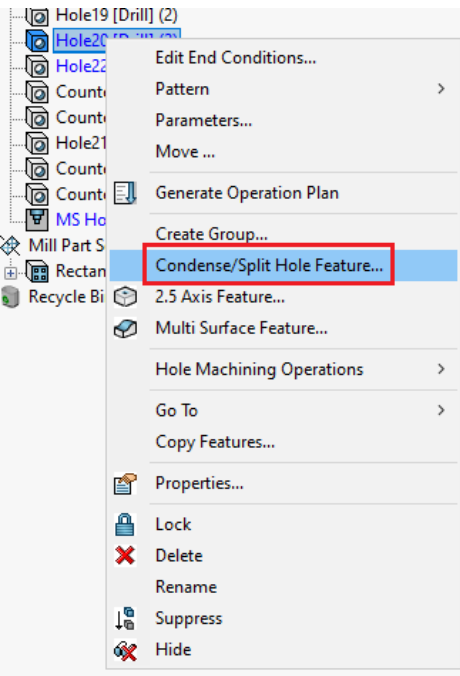


Condensed Hole Feature in Feature Tree tab

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*, 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.

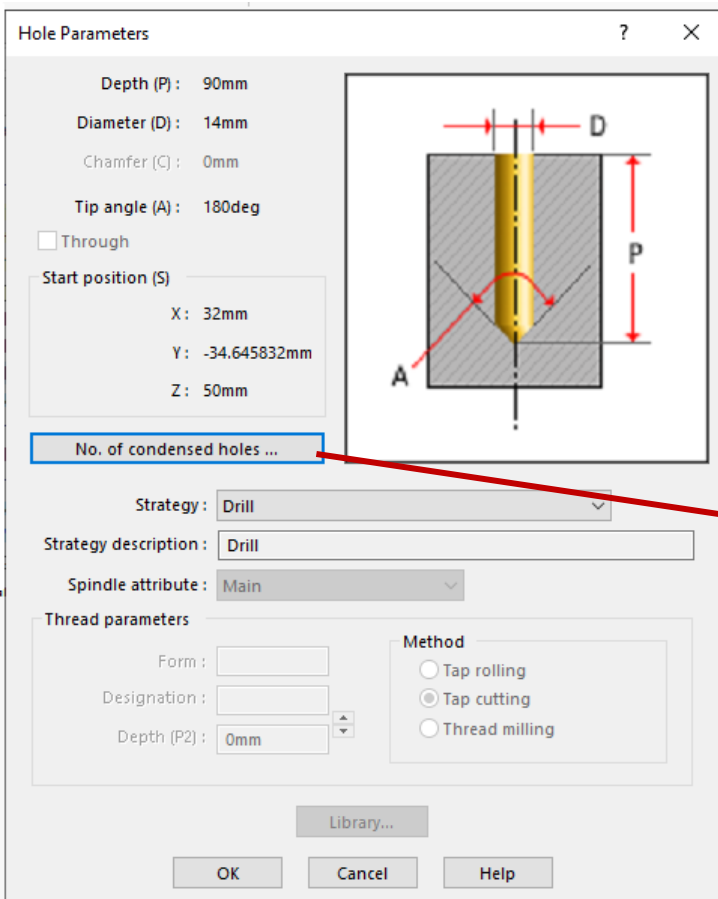
(Select the hole features to be removed from the **Selected Hole Features** list box and click the **Remove** button to separate it from the condensed hole feature. When you click the OK button, the selected features will be listed as separate hole features in the Feature tree.)

The user can add or remove multiple hole features from the list but, the basic entities defining the hole feature type (Hole feature/Counterbore/Countersink) cannot be removed from the parent feature.

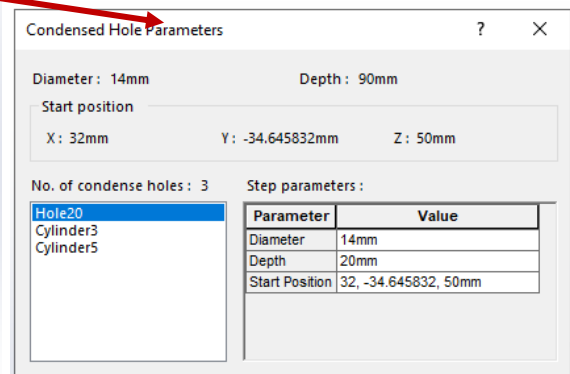
Improved: Viewing the Number of Condensed Holes in Hole Parameters Dialog Box

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 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. 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



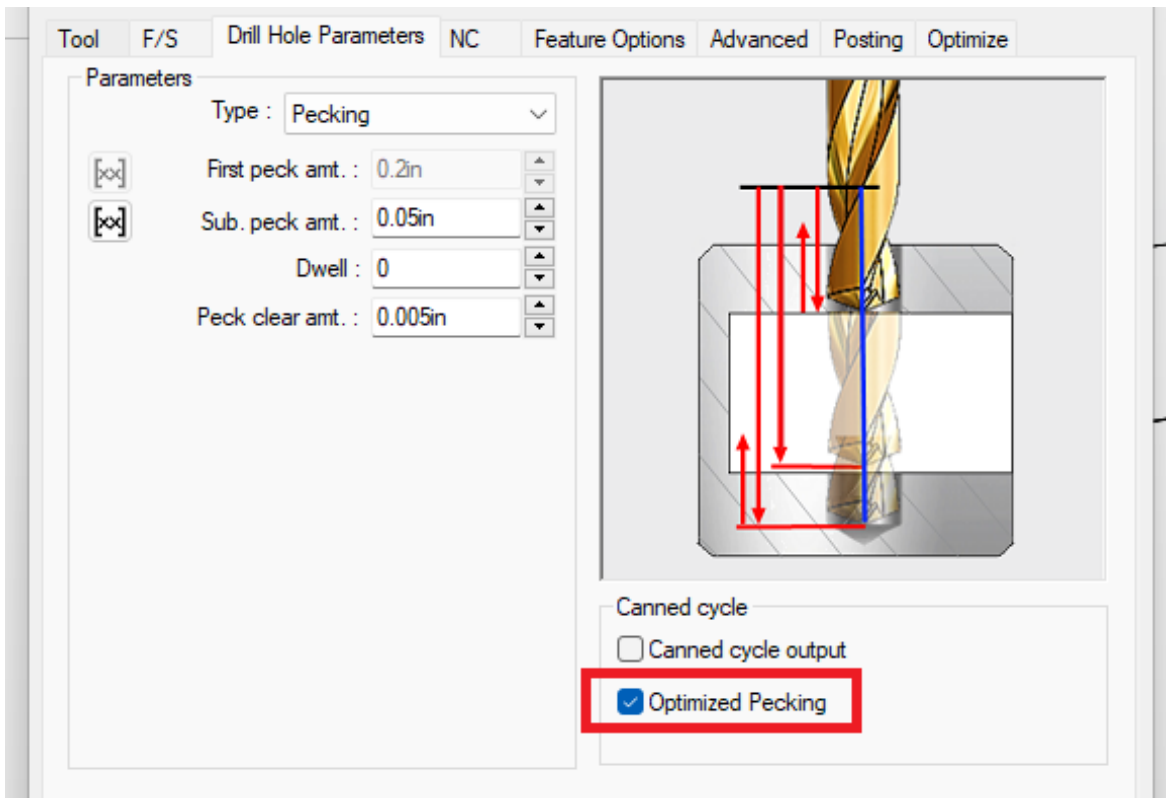
New - Optimized Pecking Option for Drill Toolpaths generated for Condensed Hole Features

Purpose:

When two or more hole features separated by a distance are combined to form a condensed hole feature then, using the option of "Optimized Pecking", the peck moves of drill toolpath can be avoided between the successive holes of the condensed hole feature

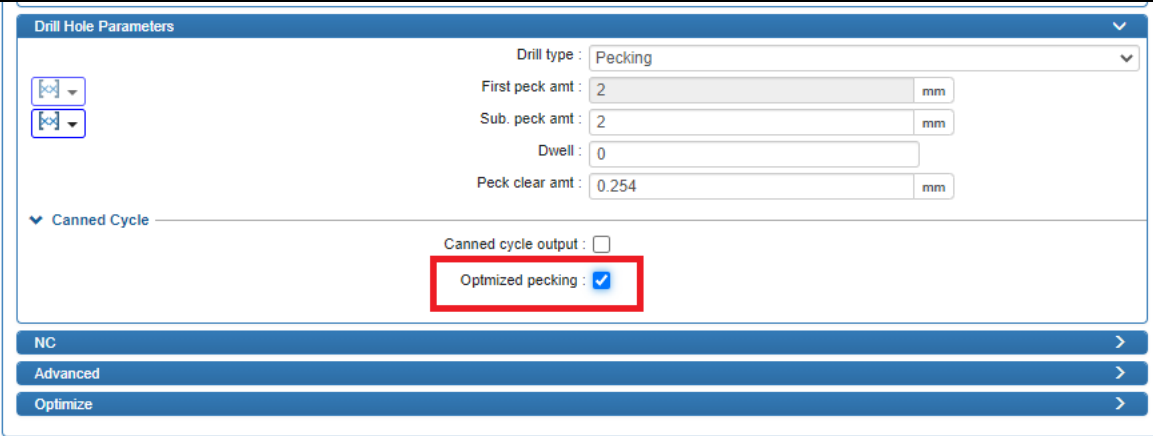
Implementation:

Condensed hole features consist of two or more coaxial cylinders of identical diameter. These cylinders are usually separated by a distance between them. In previous versions of CAMWorks, when Drill toolpaths were generated for Condensed Hole features, the toolpaths would contain peck moves even in between the consecutive hole features. From CAMWorks 2024 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, peck moves are generated from top of the hole feature till the set machining depth. Under such conditions there will be peck moves between the end of once hole feature and the start of the successive hole feature even though there will be no material to be machined. 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>Drill Toolpath when 'Optimized Pecking' option is unchecked:</p> <p>Observe that peck moves are created in the empty space.</p>	<p>Drill Toolpath when 'Optimized Pecking' option is checked:</p> <p>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>



New - Option to Generate Contour Mill Toolpaths That Machine The Features Form Bottom To Top

Purpose:

To provide the option to generate Contour Mill toolpaths which machine the features from bottom to top

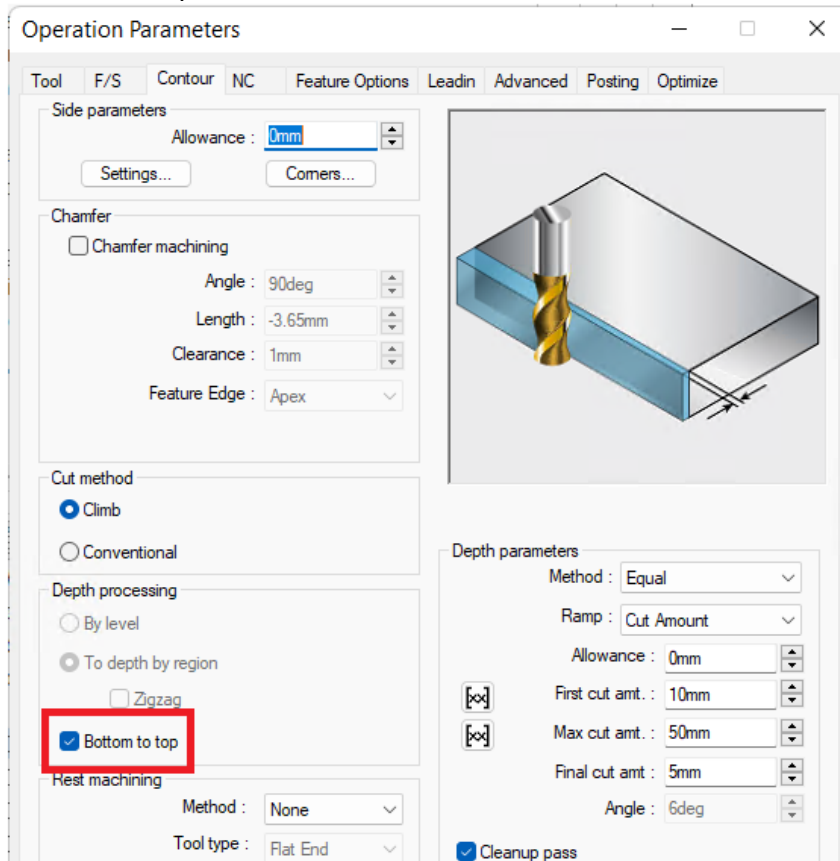
Implementation:

From **CAMWorks 2024** version onwards, the option to generate *Contour Mill* toolpaths that machine from bottom to top of 2.5 Axis Mill features has been introduced. Such Contour Mill toolpaths are ideal in the following scenarios:

- For machining tapered features
- For machining keyway slot features (Recommended tools for such features include the lollipop tool and keyway tool.)

Implementation of option for generating Contour Mill Toolpath from Bottom to Top of Feature:

For *Contour Mill* operations, a new checkbox labelled *Bottom to top* has been introduced within the *Depth Processing* group box under *Contour* tab of its *Operation Parameters* dialog box. By default, this checkbox will be unchecked. Placing a check in this checkbox will activate the option of generating the toolpath from bottom to top of the feature.



'Bottom to top' checkbox option introduced in Contour tab for Contour Mill Operations

Scenarios in which the 'Bottom to top' Option will be disabled:

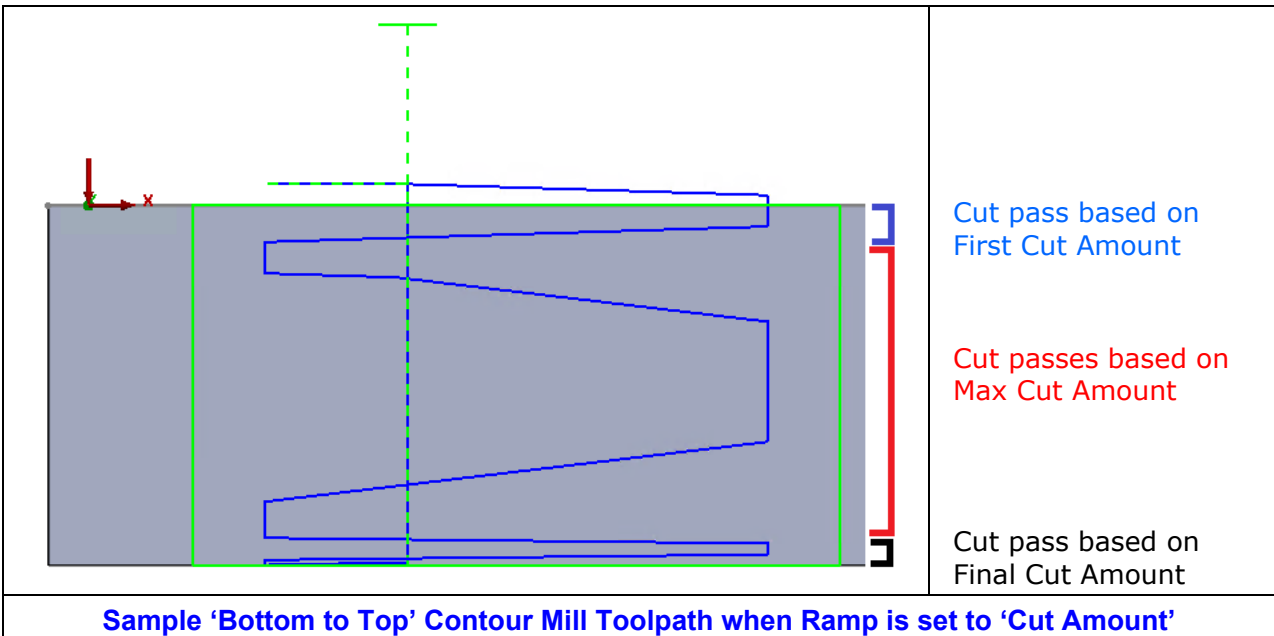
This checkbox option will be disabled for Contour Mill operations in the following cases:

- When the Contour Mill operation is used for chamfer machining (i.e., when the Chamfer machining checkbox option within the Chamfer group box under Contour tab is checked)
- When the Contour Mill operation is used for machining an Engrave feature
- When the Contour Mill operation is used for rest machining (i.e., when *Previous Tool* is assigned as the *Method* within the *Rest machining* group box under *Contour* tab is checked.)



'Bottom to top' Contour Mill Toolpath when the Ramp is set to 'Cut Amount':

- i. The tool will move in rapid motion to the bottom of the feature and cut the feature using the final cut amount.
- ii. After this final cut pass, the ramp toolpath will generate remaining cut passes from bottom to top of the feature. These cut passes will be based on the *Max Cut Amount*.
- iii. The cut pass for machining the top portion of the feature will be based on the *First cut amount*. (The *first cut amount* will always be observed at the top of the feature.)
- iv. After machining the top of the feature, the ramp toolpath will continue till the Clearance plane.
- v. From the Clearance plane, the tool will retract and plunge to the bottom of the feature to perform the Cleanup pass. (This move will occur only if the **Cleanup pass** checkbox option has been checked.)
- vi. The tool will then retract to the *Clearance plane* as defined in the NC tab.



'Bottom to top' Contour Mill Toolpath when the Ramp is set to 'OFF':

- i. The tool will rapid to the bottom of the feature and position itself at a depth just above the bottom surface of the feature equivalent to the *Final cut amount*. (The bottom-most portion of the feature with a depth equivalent to the *Final cut amount* will be machined later.)
- ii. The toolpath will commence and will make cut passes from this specific depth till a depth just below the top of the feature equivalent to First cut amount. These cut passes will be based on the Sub cut amount and *Max Cut Amount*.
- iii. The top portion of the feature will be machined based on the *First cut amount*. (The first cut amount will always be observed at the top of the feature.)
- iv. The toolpath will then go back to the bottom of the feature and make the final pass at the bottom-most portion of the feature based on *Final Cut Amount*.

Note:

If the *Final Cut Amount* is not defined, then the Contour Mill toolpath will make passes from the bottom of the feature to the top of the feature. These passes will be based on the *Sub cut amount* and *Max Cut Amount*. The pass for the topmost portion of the feature will be based on the *First cut amount*.

Setting Default Option for 'Bottom to top' Option in Technology Database for Contour Mill Operations

You can set the default setting for *Bottom to top* checkbox option in the *Technology Database* using the Contour Mill Form for *Contour Mill Operations*.



The screenshot displays the CAMWorks 2024 Technology Database interface. On the left is a navigation menu with icons for Mill, Turn, Mill-Turn, EDM, Mill Tooling, Turn Tooling, Feed / Speed, Settings, and About. The main window is titled 'Mill > Default Operation Parameters' and shows a table of 'Contour Mill' parameters. The table has columns for Id, Name, Description, and Default. The first row (Id: 67) is selected. To the right of the table is a detailed form for the 'Contour Mill' operation. The form includes fields for ID (67) and Node description (CONTOUR MILL). Under the 'Depth processing' section, there are checkboxes for 'Zigzag' and 'Bottom to top'. The 'Bottom to top' checkbox is highlighted with a red rectangle.

Id	Name	Description	Default
67	Default	Default - Inch	✓
118	Contour machining	Contour machining only	
119	Contour and floor mac...	With bottom finish	

'Bottom to top' Option in Contour Tab of Contour Mill Form on Technology Database



Turn/Mill-Turn

New - Highlight Recognized/Unrecognized Faces of the Turn Part

Purpose:

- To provide the option to highlight faces on the Turn part model in the graphics area that have been 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**.

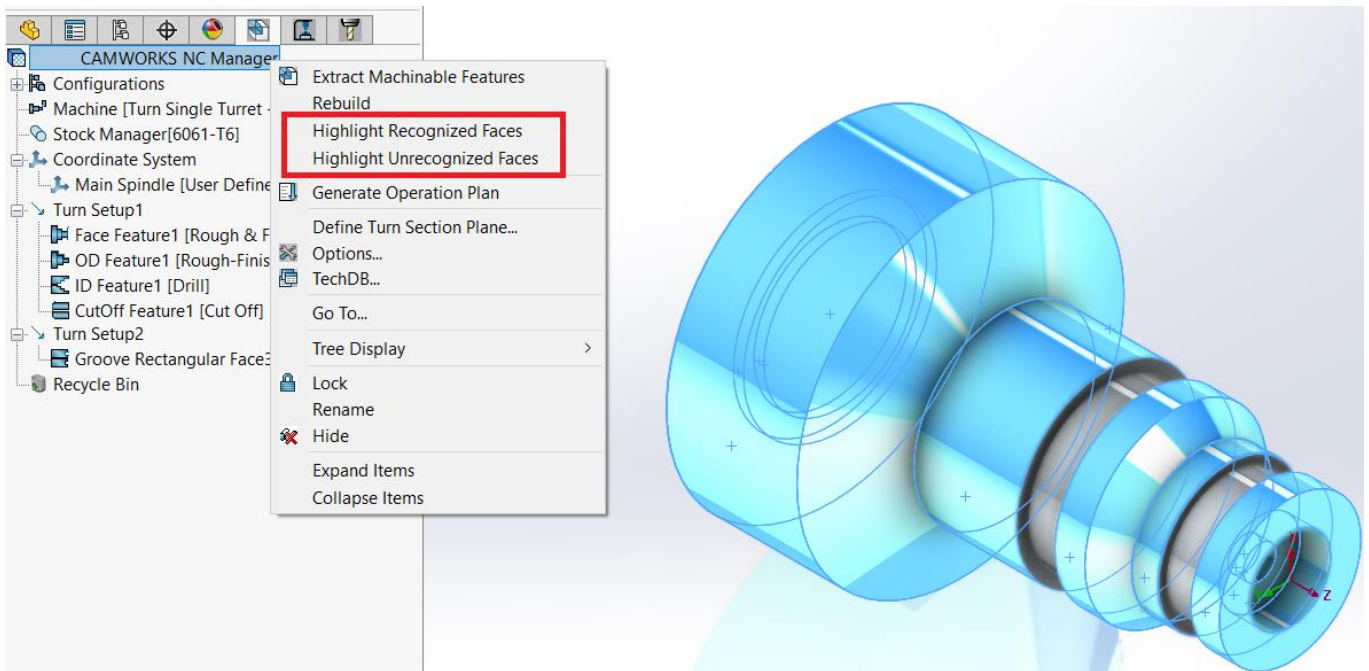
Implementation:

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 Faces** 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 Faces** option in the context menu of CAMWorks NC Manager node in the Feature tree.

From CAMWorks 2024 version onwards, this functionality has been extended to parts machined on Turn machines. 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 Faces** and **Highlight Unrecognized Faces** for the CAMWorks NC Manager node in the Feature tree.

Note:

- This functionality to highlight faces recognized or left unrecognized is 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 Faces** and **Highlight Unrecognized Faces** 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 Main and Sub Spindle Coordinate System (FCS) in Turn Assembly Mode Using SOLIDWORKS Coordinate Systems

Purpose:

To provide the ability to define **Main and Sub Spindle Coordinate Systems** for Turn Assemblies in CAMWorks Turn mode using SOLIDWORKS coordinate systems.

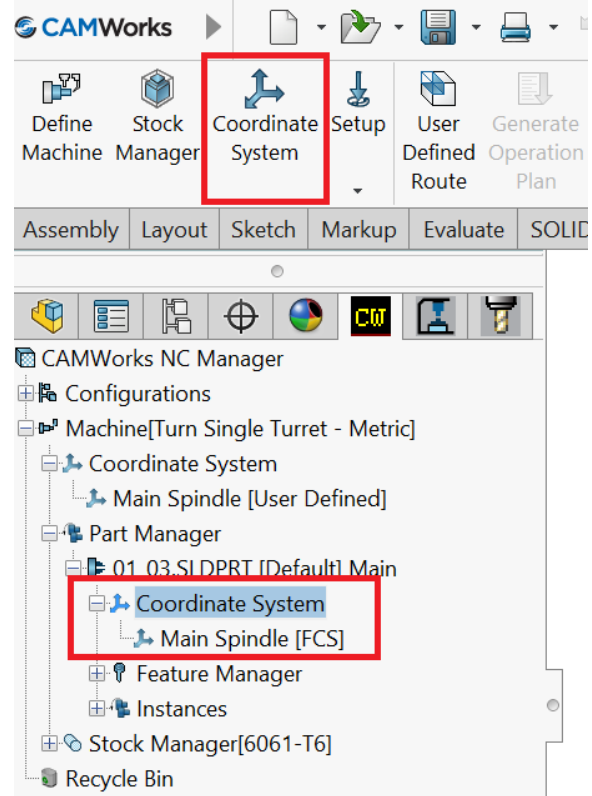
Implementation:

In previous versions of CAMWorks, the option to define Main and Sub Spindle Coordinate system using the SOLIDWORKS coordinate system was available only in part mode for Turn machines. From CAMWorks 2024 version onwards, this functionality has been extended to Turn Assembly mode.

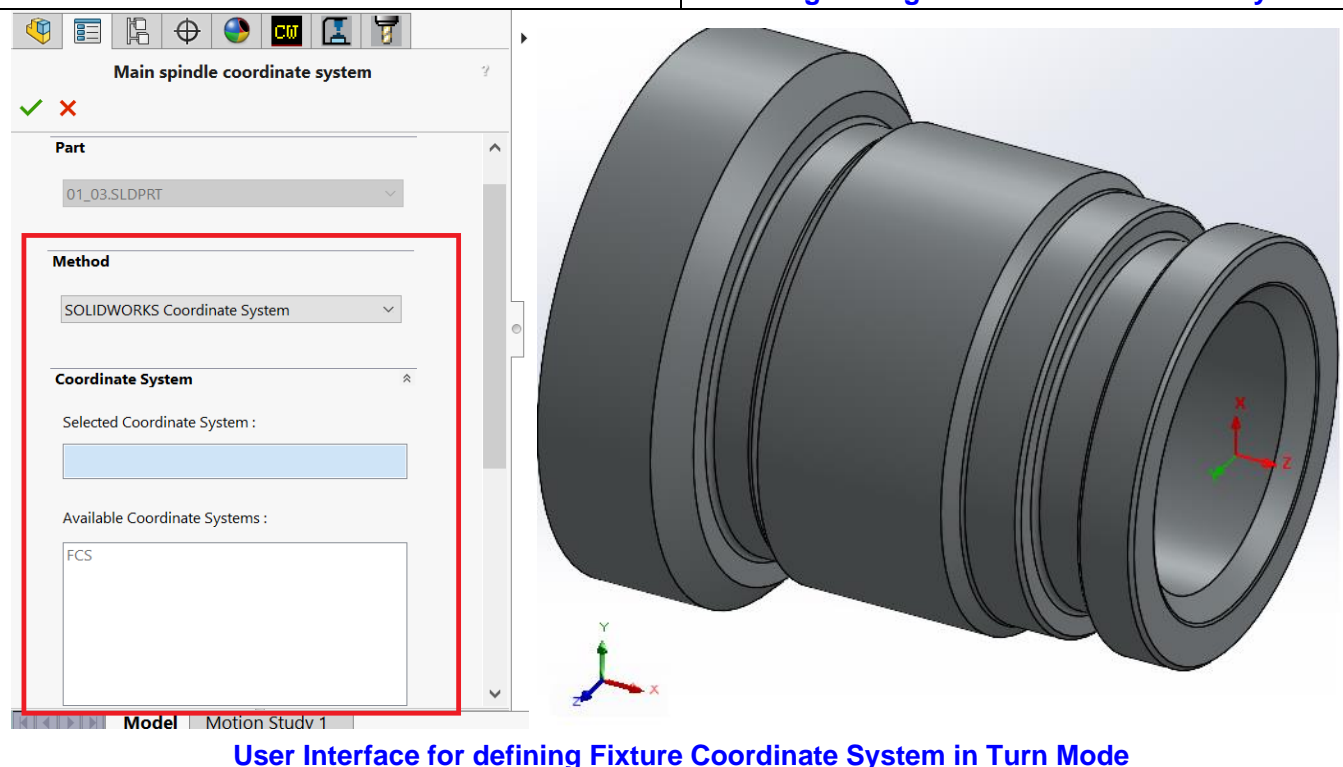
While defining the Coordinate system for Machine and Parts in the Assembly mode, users can now select the available SOLIDWORKS Coordinate Systems.

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 - Turn Tool Block Library In TechDB

Purpose:

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

Implementation:

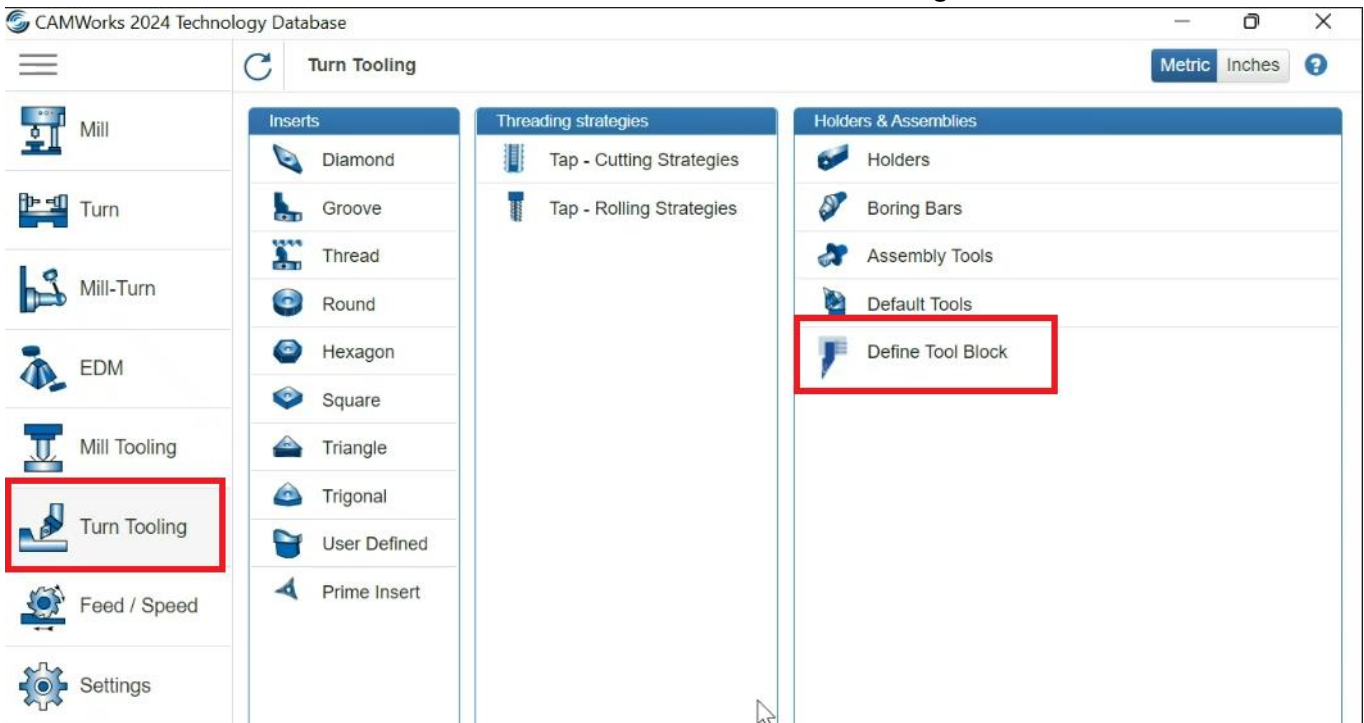
The tool blocks have been playing a key role in verifying the toolpaths in CAMWorks Virtual Machine. These tool blocks have to be created using the **CAMWorks User Defined Tool Block** utility.

The Tool Blocks have to be created based on the machine and stored under tool block folder of the respective virtual machine folder. And In previous versions of *CAMWorks*, the tool block for any Turn tool could be assigned only in the **Operation Parameters** dialog box. 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* version onwards, user can create a single list of the tool blocks in the TechDB. The list will be auto populated using the simulation machine path defined in the settings tab of TechDB. Whenever new tool blocks are added to any of the machines or existing tool blocks are deleted, then, the list of the tool blocks can be automatically updated. The desired tool block can be added to the turn tool while defining the Tool Crib in TechDB.

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.

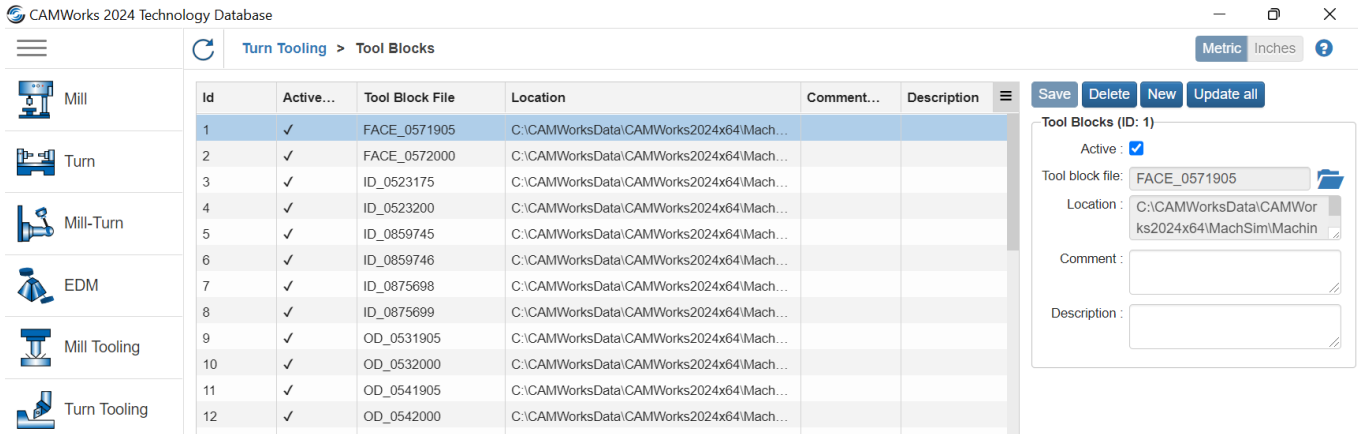
Table of Tool Blocks

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 Simulation machine path of CAMWorks Virtual machines 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. The file path to that specific *TBDINF file will be displayed in the **Location** field within this form.



Id	Active...	Tool Block File	Location	Comment...	Description
1	✓	FACE_0571905	C:\CAMWorksData\CAMWorks2024x64\Mach...		
2	✓	FACE_0572000	C:\CAMWorksData\CAMWorks2024x64\Mach...		
3	✓	ID_0523175	C:\CAMWorksData\CAMWorks2024x64\Mach...		
4	✓	ID_0523200	C:\CAMWorksData\CAMWorks2024x64\Mach...		
5	✓	ID_0859745	C:\CAMWorksData\CAMWorks2024x64\Mach...		
6	✓	ID_0859746	C:\CAMWorksData\CAMWorks2024x64\Mach...		
7	✓	ID_0875698	C:\CAMWorksData\CAMWorks2024x64\Mach...		
8	✓	ID_0875699	C:\CAMWorksData\CAMWorks2024x64\Mach...		
9	✓	OD_0531905	C:\CAMWorksData\CAMWorks2024x64\Mach...		
10	✓	OD_0532000	C:\CAMWorksData\CAMWorks2024x64\Mach...		
11	✓	OD_0541905	C:\CAMWorksData\CAMWorks2024x64\Mach...		
12	✓	OD_0542000	C:\CAMWorksData\CAMWorks2024x64\Mach...		

Form fields on the right:


- Active:
- Tool block file: FACE_0571905
- Location: C:\CAMWorksData\CAMWorks2024x64\MachSim\Machin...
- Comment:
- Description:

Turn Tool Blocks User Interface

Assigning Tool Blocks to Turn Tool Cribs in TechDB

In the Turn and Mill Turn Tool Cribs user interface of TechDB, the option to assign Tool Blocks to Turn tools is available in the form of the **Tool Block** parameter. Use this parameter to browse, select and assign the desired tool block for the current station of the active tool crib.

Steps to assign a tool block for a Turn Tool in Tool Crib

1. Click on the **Select tool block** 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** parameter 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.

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. 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.

What's New in CAMWorks 2024 SP0



CAMWorks 2024 Technology Database

Turn > Tool Cribs

Metric Inches

New Copy Edit Delete

Select Tool Crib: Tool Crib 2 Rear (Metric) Total stations: 12 Define mill tools using tool assembly

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

Add Save Copy Delete

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

CAMWorks 2024 Technology Database

Turn > Tool Cribs

Metric Inches

Select

Tool Blocks

Id ...	Active ...	Tool Block File	Comment	Description ...
60	<input checked="" type="checkbox"/>	OD_0532000		
61	<input checked="" type="checkbox"/>	OD_0541905		
62	<input checked="" type="checkbox"/>	OD_0542000		
63	<input checked="" type="checkbox"/>	OD_8694753		
64	<input checked="" type="checkbox"/>	OD_BGL03175		
65	<input checked="" type="checkbox"/>	OD_BGL03200		
66	<input checked="" type="checkbox"/>	FACE_NKM0021620		
67	<input checked="" type="checkbox"/>	FACE_NKM0181905		
68	<input checked="" type="checkbox"/>	FACE_NKM0182000		
69	<input checked="" type="checkbox"/>	ID_NKM0113221		

Save Delete New Update all

Tool Blocks (ID: 76)

Active:
 Tool block file: OD_NKM0162000
 Comment:
 Description:

User Interface displayed on clicking the Select tool block button adjacent to Tool Block field



Improved - Define Turn Operations on the Sub Spindle Setup at the start of the tree

Purpose:

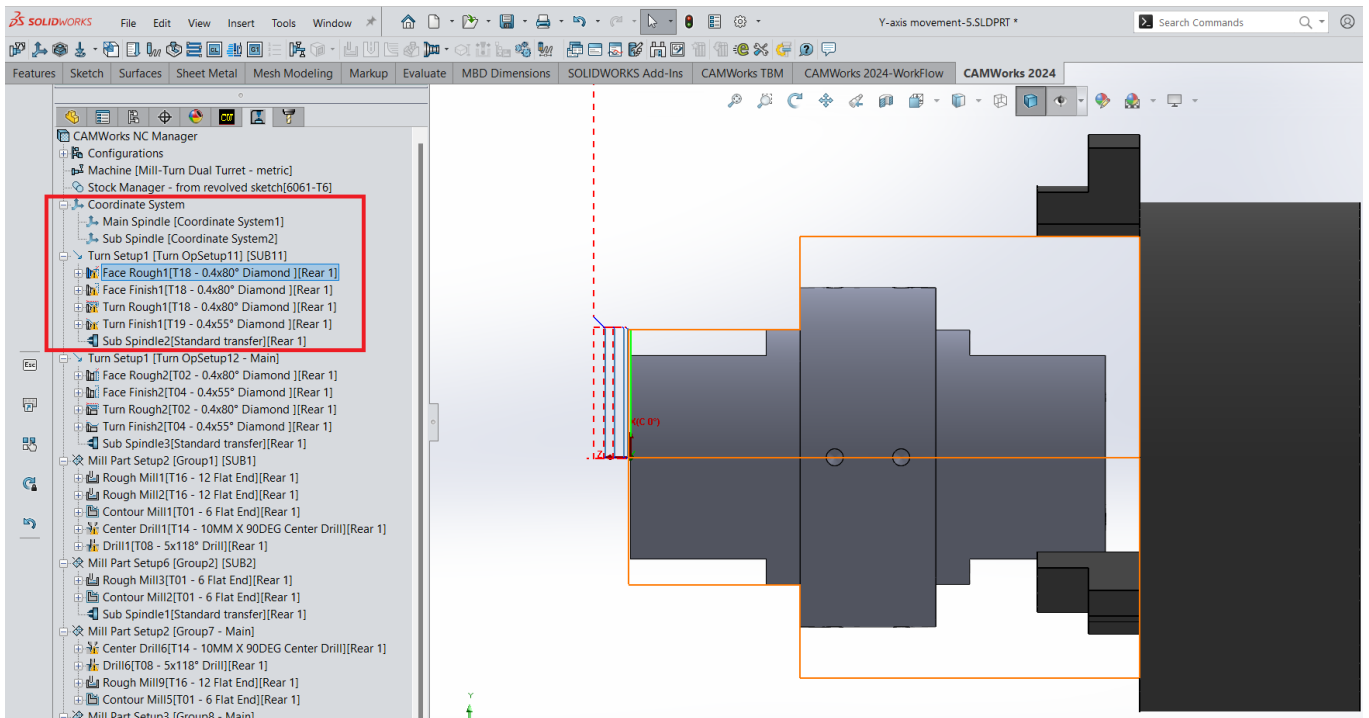
To allow the user to define sub spindle setup and operations at the start of the tree so that user can start programming by clamping the part in the sub spindle

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. Users can create Sub spindle setups and operations at the start of the tree so that the features to be machined in the sub spindle are programmed at the beginning. Later if required, the user can add sub spindle operation to transfer the part to the main spindle and continue the programming. 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:

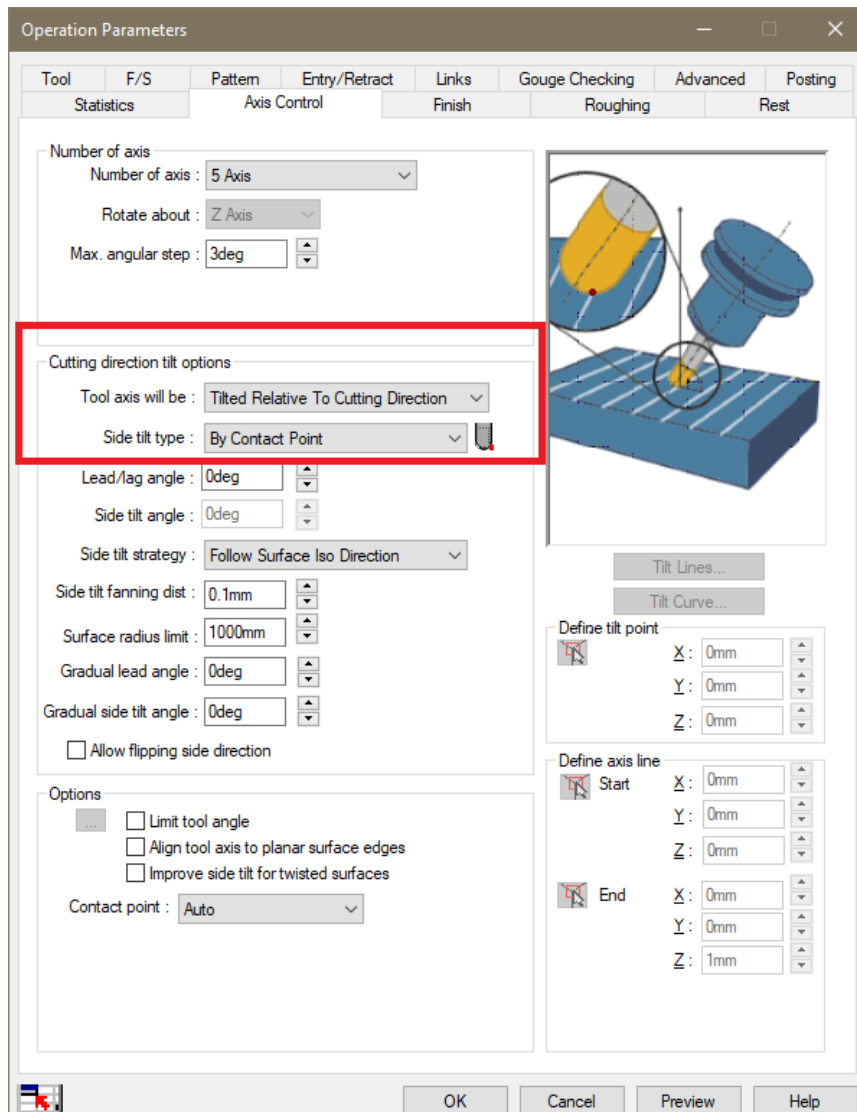
Option to tilt tool by specifying a contact point for the cutting direction for Multiaxis Mill operations

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 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:

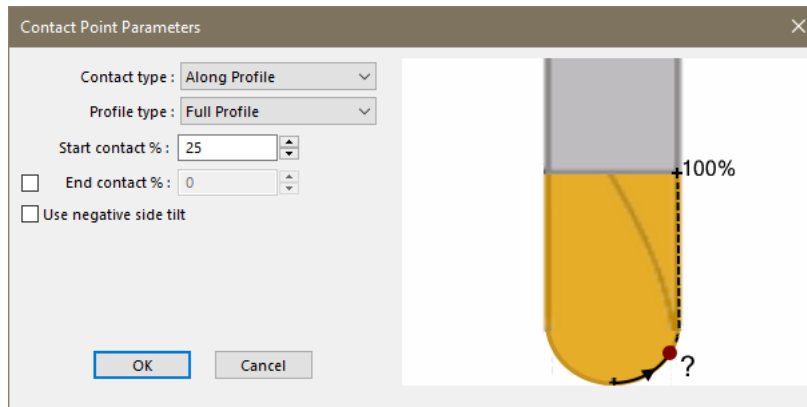
- **By Angle**

This is the existing strategy that was available in previous versions of CAMWorks too as explained in the previous paragraph. From *CAMWorks 2024* 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* 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.



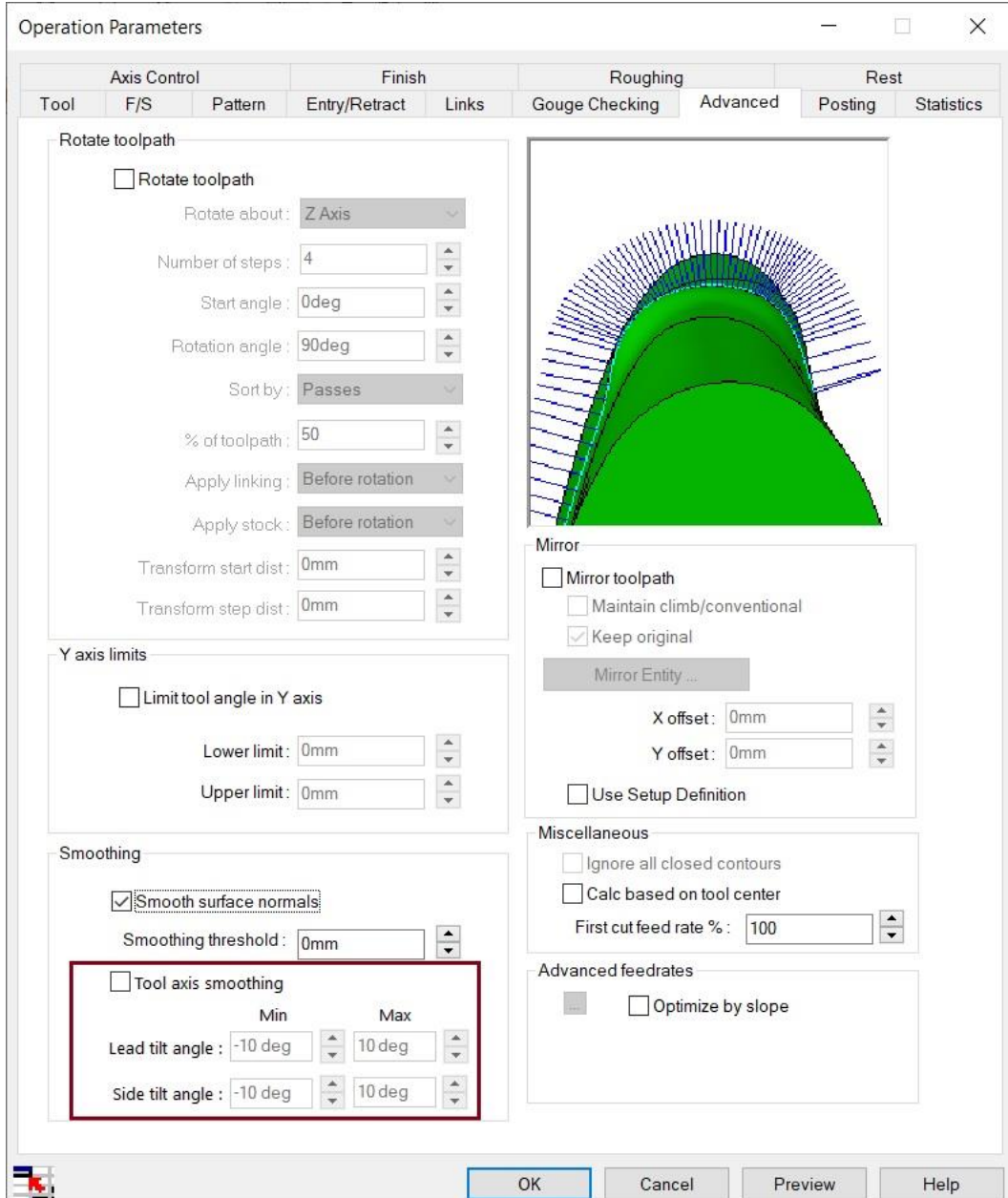
New - Tool Axis Smoothing Functionality for 5 Axis Mill Operations

Purpose:

Option to enable Tool Axis Smoothing functionality for 5 Axis Mill Operations

Implementation:

For Multiaxis (5 Axis) Mill operations, under the **Advanced** tab, new parameters for tool axis smoothing functionality have been introduced.



New 'Tool axis smoothing' parameters under Advanced Tab of 5 Axis Mill Operations

The Tool axis smoothing functionality will be enabled when you place a check in the **Tool axis smoothing** checkbox within the **Smoothing** group box. Checking this box will enable the **Lead tilt angle** and **Side tilt angle** parameters. Use these parameters to edit the tilt angle range for the tool axis smoothing function.



Simulation

Improved - Dialog Box displaying Legend during Toolpath Simulation

Purpose:

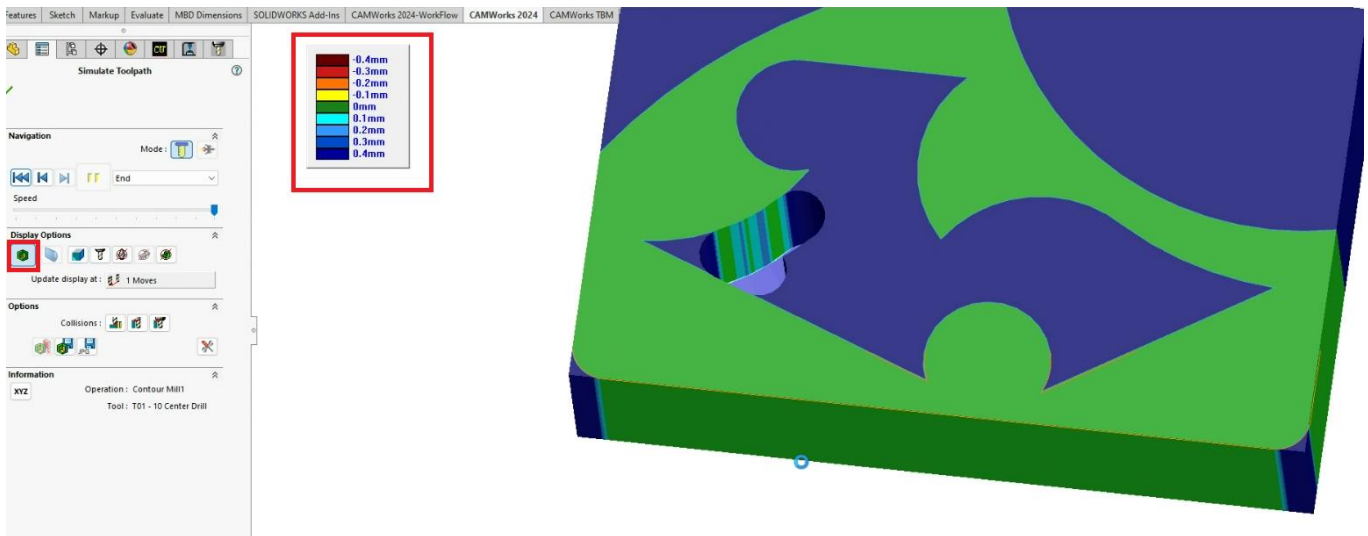
To provide a dockable dialog box indicating the legend for graphical comparison of the machined part and the design part during toolpath simulation

Implementation:

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

In previous versions of *CAMWorks*, a legend at the top right corner of the graphics area listed these assorted colors and the corresponding deviation indicated by those colors. The legend was always displayed in the upper left corner and could not 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 machined part and design part displayed in a dockable window



TechDB

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

Purpose:

To provide the option to assign a 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.

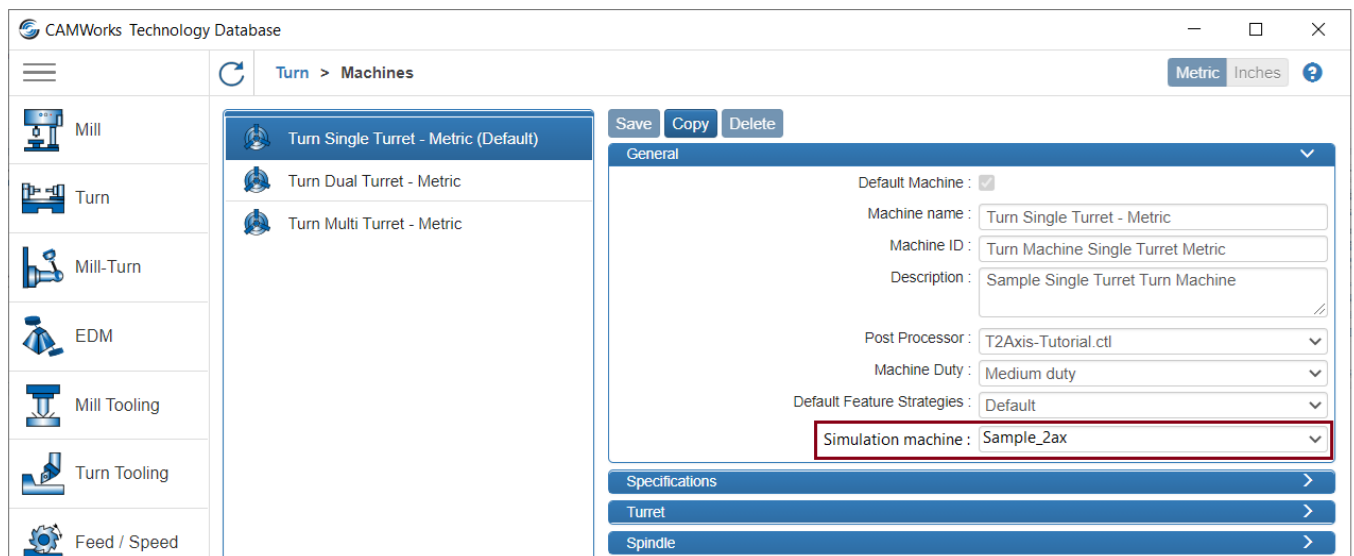
This dropdown field is used to assign the default virtual simulation machine that will be associated with the selected machine. To assign another simulation machine, click on the **Simulation machine** field. A dropdown list of all the simulation machines compatible with the currently selected machine 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\CAMWorks2024x64\MachSim\Machines

This is the default path set under "**Simulation machine path**" under TechDB settings. Under default TechDB settings, for a selected machine in the **Machine** 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 type.



Assigning Simulation Machine within Machine User Interface of TechDB

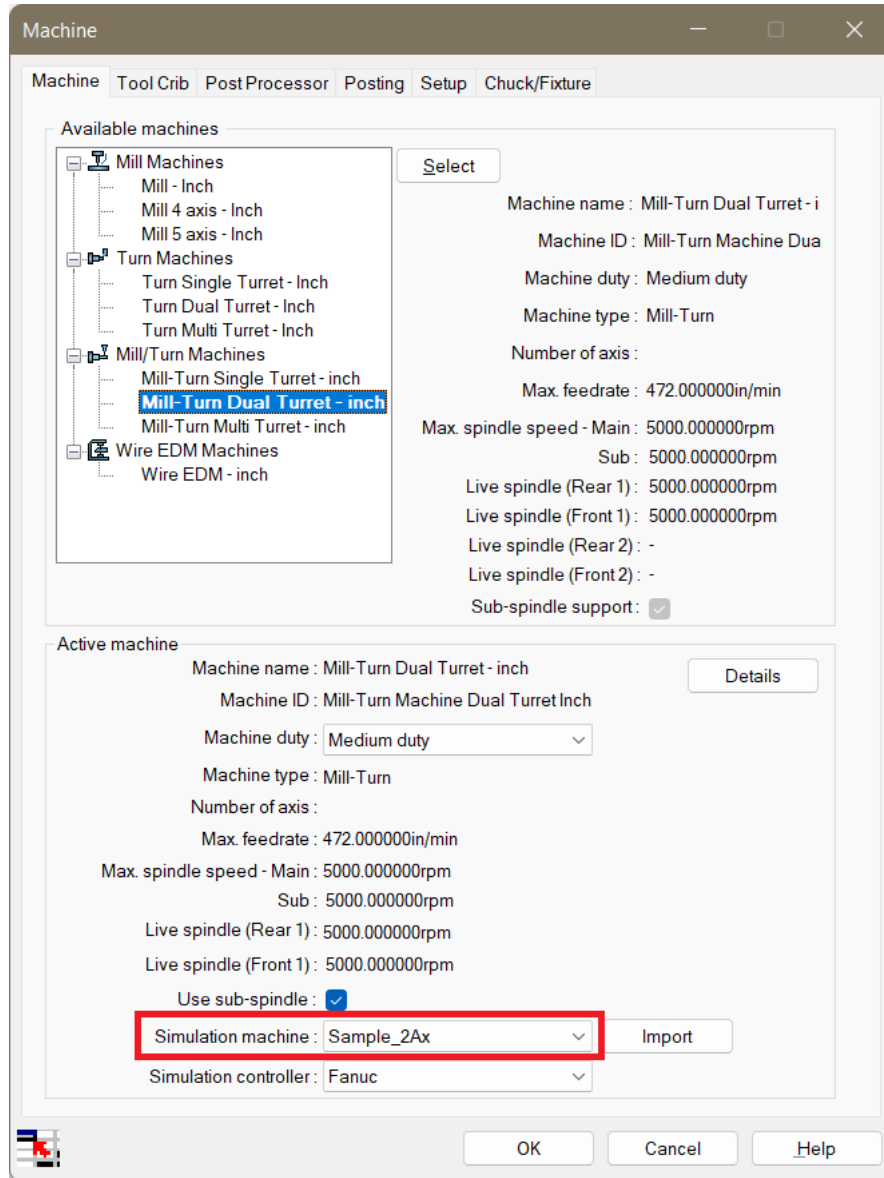
Populating 'Simulation Machine' Dropdown list with Customized Simulation Machines

All the required Virtual Simulation Machine data should be stored under the path set as "Simulation Machine Path" so that the **Simulation Machine** dropdown list under **Machine** tab of **Machine** dialog box is populated with relevant machines.



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.



Simulation Machine Dropdown list under Machine tab of Machine Dialog Box

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.