What's New

in

CAMWorks 2021

October 2021

Table of Contents

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

What's New in CAMWorks 2021 – SP4

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2022 - SOLIDWORKS 2021 - SOLIDWORKS 2020 - CAMWorks Solids 2021 - CAMWorks Solids 2020 **Note:** CAMWorks Solids 2022 will be supported once released.
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] **Note:** CAMWorks 2021 is supported only on 64-bit Operating systems.

Resolved CPRs document

Purpose:	The Resolved CPR (*CAMWorks Problem Report*) document has been updated to report the software errors that have been resolved in the current Service Pack (SP4).
Implementation:	To view the document, select: *Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.*

What's New in CAMWorks 2021 – SP3

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2021 - SOLIDWORKS 2020 - CAMWorks Solids 2021 - CAMWorks Solids 2020
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] **Note:** CAMWorks 2021 is supported only on 64-bit Operating systems.

Resolved CPRs document

Purpose:	The Resolved CPR (*CAMWorks Problem Report*) document has been updated to report the software errors that have been resolved in the current Service Pack (SP3).
Implementation:	To view the document, select: *Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.*

What's New in CAMWorks 2021 – SP2

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2021 - SOLIDWORKS 2020 - CAMWorks Solids 2021 - CAMWorks Solids 2020
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] **Note:** CAMWorks 2021 is supported only on 64-bit Operating systems.

Resolved CPRs document

Purpose:	The Resolved CPR (*CAMWorks Problem Report*) document has been updated to report the software errors that have been resolved in the current Service Pack (SP2).
Implementation:	To view the document, select: *Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.*

What's New in CAMWorks 2021 – SP1

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2021 - SOLIDWORKS 2020 - CAMWorks Solids 2021 - CAMWorks Solids 2020
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] **Note:** CAMWorks 2021 is supported only on 64-bit Operating systems.

Resolved CPRs document

Purpose:	The Resolved CPR (*CAMWorks Problem Report*) document has been updated to report the software errors that have been resolved in the current Service Pack (SP1).
Implementation:	To view the document, select: *Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.*

New - Tutorial Document on Assembly Mode of Turn/Mill-Turn Module

Purpose: A tutorial document illustrating how the Assembly Mode of Turn/Mill-turn module of CAMWorks can be used for machining assemblies
Implementation: Assembly mode support Turn/Mill-Turn module was introduced in *CAMWorks 2021 SP0* version. A tutorial document with illustrative examples on this Turn/Mill-Turn Assembly mode will now be available from *CAMWorks 2021 SP1* version onwards. Once ***CAMWorks*** is loaded as an Add-in within *SOLIDWORKS/ CAMWorks Solids*, this tutorial document can be accessed from the *SOLIDWORKS/CAMWorks Solids* ***Help*** menu by selecting ***CAMWorks 2021>>Tutorials***. Executing this command will display a window listing all the CAMWorks documents in PDF format. Double-click on the document named ***Assembly_Mode_of_Turn_MillTurn_Tutorial.pdf***. This document contains two tutorials illustrative examples of how to use the *Assembly* mode of the Turn/Mill-Turn module of *CAMWorks*.

What's New in CAMWorks 2021 – SP0

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2021 - SOLIDWORKS 2020 - CAMWorks Solids 2020 **Note:** CAMWorks Solids 2021 will be supported when released.
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] **Note:** CAMWorks 2021 is supported only on 64-bit Operating systems.

Resolved CPRs document

Purpose:	The Resolved CPR (*CAMWorks Problem Report*) document has been updated to report the software errors that have been resolved in the current Service Pack (SP0).
Implementation:	To view the document, select: *Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.*

Mill

New - Prompt for Rebuild of CAM data when Stock is Modified

Purpose:

Option to rebuild features, operations, and toolpath data whenever there is a change to the stock parameters

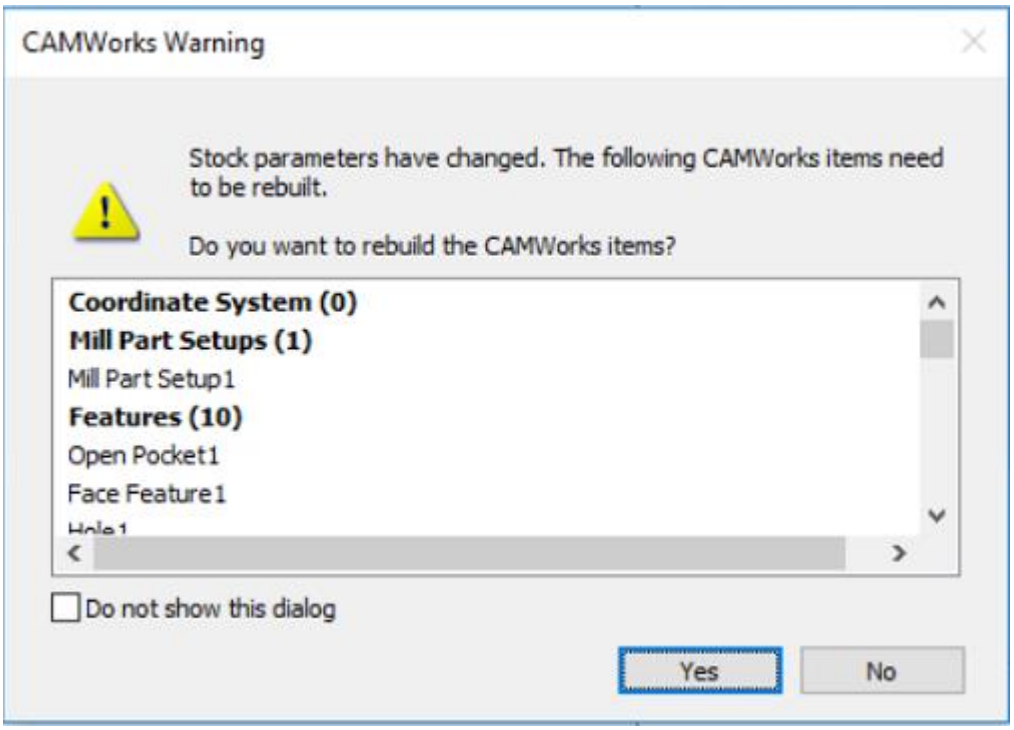
Implementation:

For Mill and Turn part models, many features and operations parameters can be associated with the assigned stock parameters. Any changes to the stock can affect these parameters and consequently, the generated toolpaths.

In the Turn module of CAMWorks, whenever the stock associated with a Turn part model is modified, a warning message that prompts users to regenerate the toolpaths is displayed.

From *CAMWORKS 2021* version onwards, this functionality has been extended to the Mill and Mill-Turn module too. In a Mill part, whenever the stock parameters are modified after the generation of features, operations, and toolpaths, then a warning message prompting user to rebuild the data will be displayed. The warning message box will list all the *CAMWORKS* entities in the part/assembly that will be affected by the change in stock. Such entities include Coordinate Systems, Mill Part Setups, Features and Operations. (These items are listed in the *CAMWORKS* Feature tree and Operation tree.)

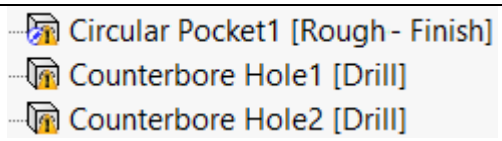
- Click 'Yes' within this warning message box to rebuild the data.
- Click 'No' within this warning message box to ignore the warning. Data will not be rebuilt.
- If you do not wish to view this warning every time you change the stock parameters, place a check in the ***Do not show this dialog*** checkbox option within this warning message.



Sample Warning Message Prompting Users to Rebuild Data after Stock Parameters are Modified

Impact when you select 'No' within the Message Box

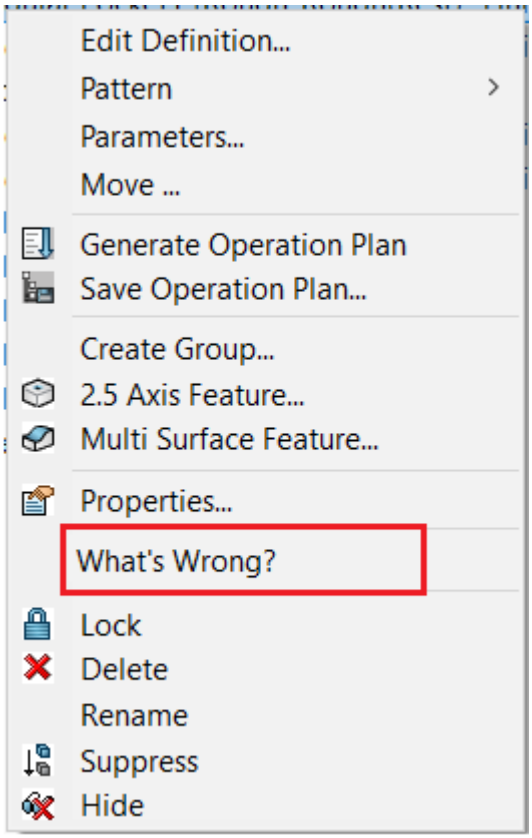
If you select 'No', the *CAMWORKS* entities in the Feature tree and Operation tree affected by the change in stock parameters will be flagged with a yellow exclamation mark appearing over their respective icons.



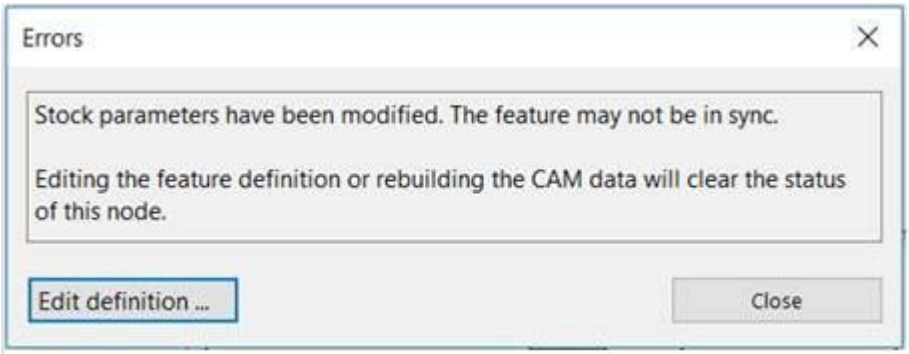
Feature node items in Feature Tree flagged with Exclamation Mark on selecting 'No/ in Warning Message Box

- For a flagged feature node item, this exclamation mark will disappear only after you edit the definition for that item or perform rebuild to sync the data with the stock parameters. The menu option to rebuild data will be available in the context menu of the *CAMWorks NC Manager* node.
- For a flagged operation node item, this exclamation mark will disappear only after you regenerate the toolpaths.

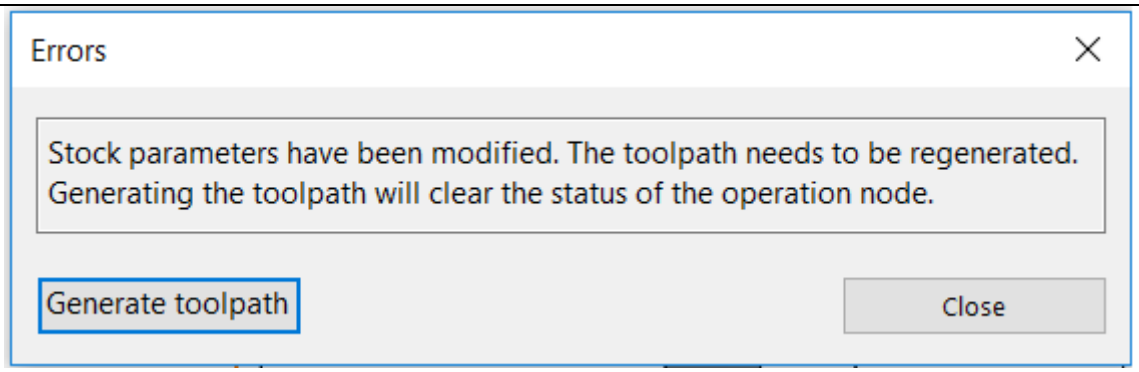
For flagged Feature nodes and Operation nodes in the Feature tree and Operation tree respectively, the menu option ***What's Wrong?*** will be available in their context menus if they remain flagged.



'What's Wrong' option in Context Menu of Operation Node



Error message displayed when 'What's Wrong' option in Context Menu for a Flagged Feature Node is selected

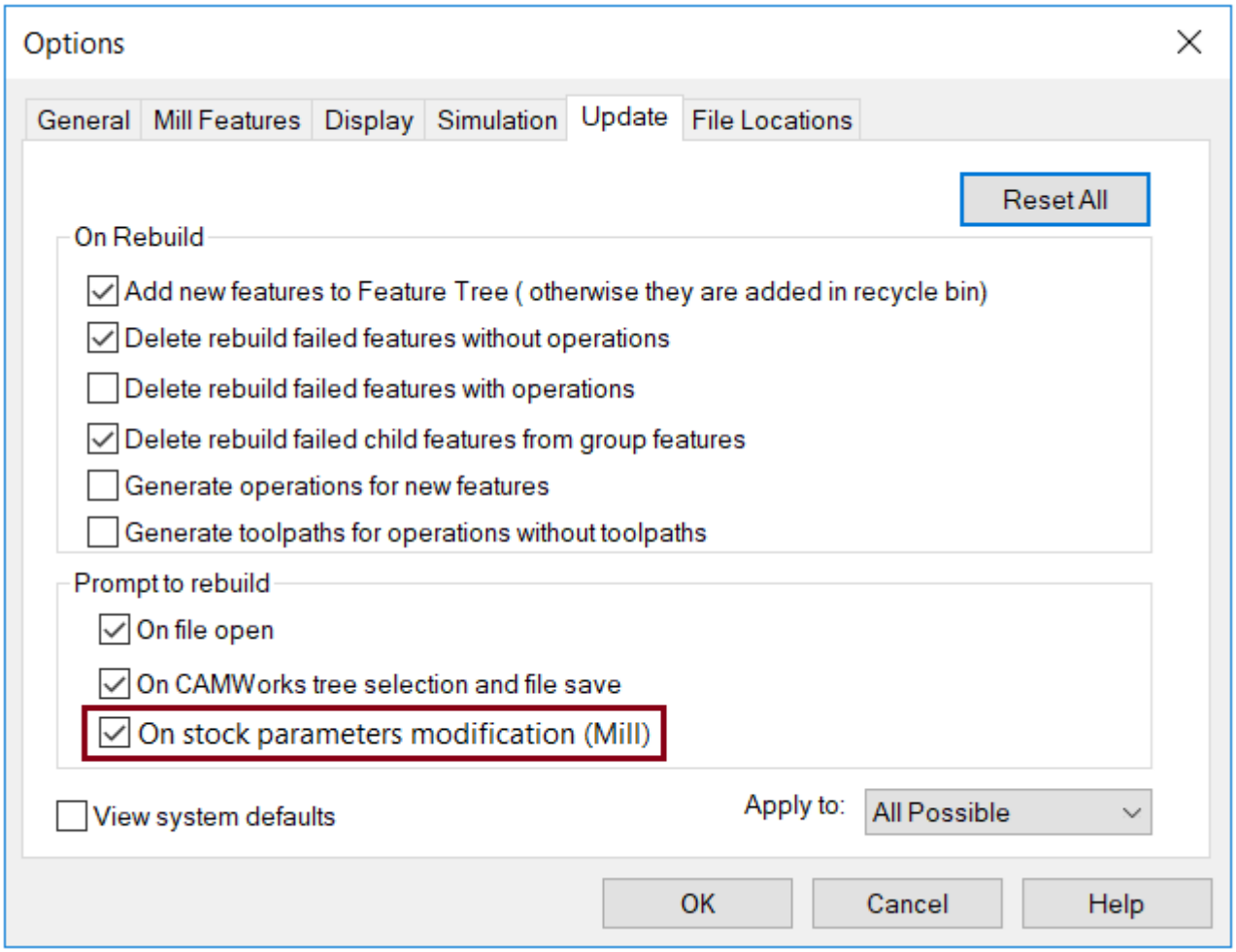


Error message displayed when 'What's Wrong' option in Context Menu for an Flagged Operation Node is selected

Controlling the Display of the Warning Message

The display of this warning message is controlled by the ***On Stock Parameters Modification (Mill)*** checkbox option in the **Prompt to rebuild** group box on the ***Update*** tab of the ***Options*** dialog box. (***Options*** command to display this dialog box is available on the *CAMWorks* Command Manager.)

By default, this checkbox option will be checked. When you place a check in the ***Do not show this dialog*** checkbox within the Warning Message box, the status of the ***On Stock Parameters Modification (Mill)*** checkbox option will change to unchecked state. You can resume display of the warning message at any point of time by placing a check back in the ***On Stock Parameters Modification (Mill)*** checkbox option.



Option in Updates Tab of Options Dialog Box

New - Option for Defining Cylindrical Stock Type in Mill Mode

Purpose:

To provide a functionality whereby users can define Cylindrical stock for Mill Parts

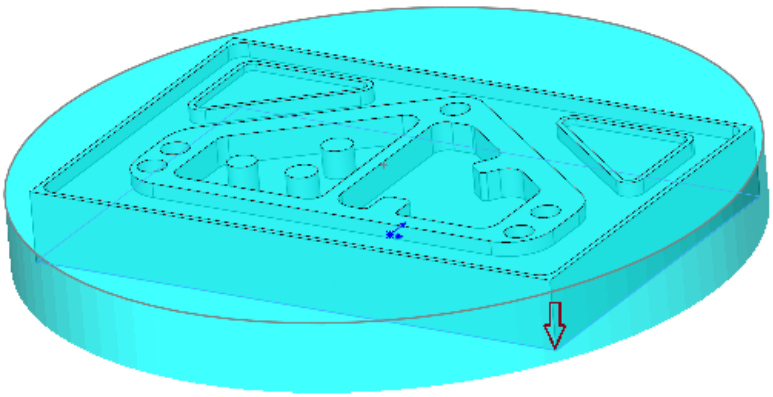
Implementation:

In previous versions of *CAMWORKS*, the stock type for a Mill part model/assembly could be defined from a bounding box, extruded sketch, STL file or another part file. The option to directly define a cylindrical stock type was not available.

From *CAMWORKS 2021* version onwards, the option to define a stock of cylindrical shape will be available in the *Stock Manager* dialog box user interface for both Part mode and Assembly mode.

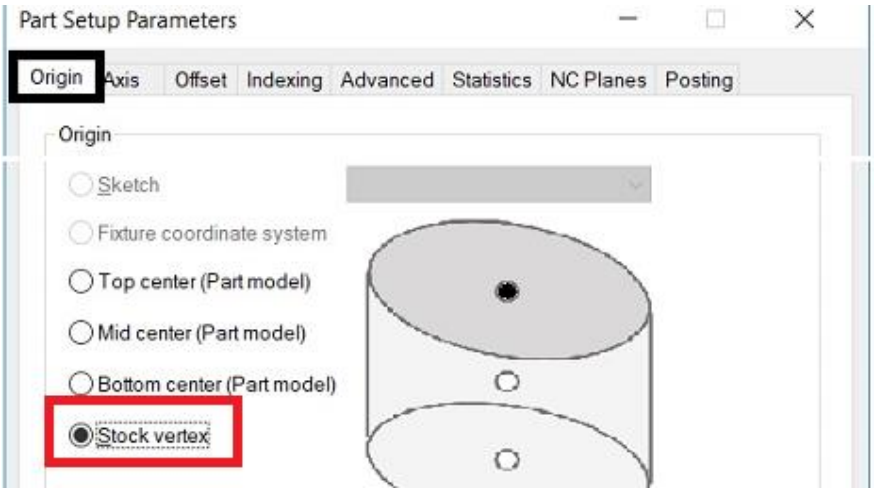
When the Cylindrical Stock type option is selected, users will have the following parameters to define the stock:

- Centre axis of the stock (Default Centre axis will be the Z axis of the assigned Fixture Coordinate System)
- Origin of the stock (Default origin will be the centroid of the mill part model)
- Diameter of Cylinder (Default diameter will be the minimum diameter based on the corresponding dimension of the part model such the part fits within it; refer image below.)
- Length of cylinder (Default length will be the minimum length based on the corresponding dimension of the part model such the part fits within it; refer illustrative image below.)
- Offsets, if any.

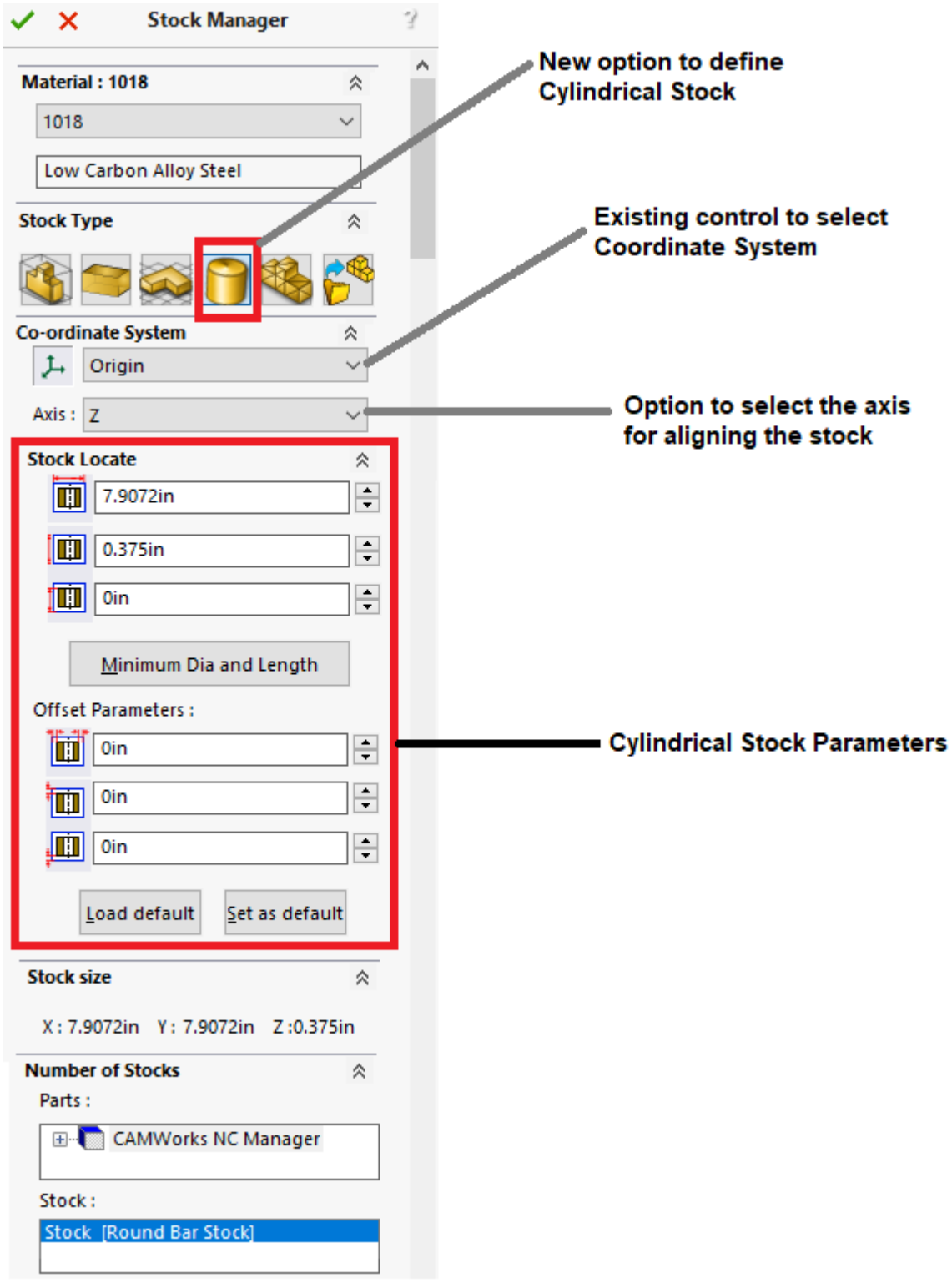


Mill Part Model with Cylindrical Stock of Minimum Diameter and Length
Note that the Z Axis of Cylindrical Stock parallel to the Mill Part Setup

If the axis of the cylindrical stock is parallel to the Z axis of a Mill Part Setup for the part model, then you can define the origin of the Mill Part Setup using one of the three central vertices (Top Center, Mid Center and Bottom Center) of the cylindrical stock. This option is available in the ***Origin*** tab of the ***Part Setup Parameters*** dialog box.



Options to choose Origin of Mill Part Setup in Origin tab of Part Setup Parameters Dialog Box



Stock Manager Dialog Box UI Parameters when Cylindrical Stock Type is selected

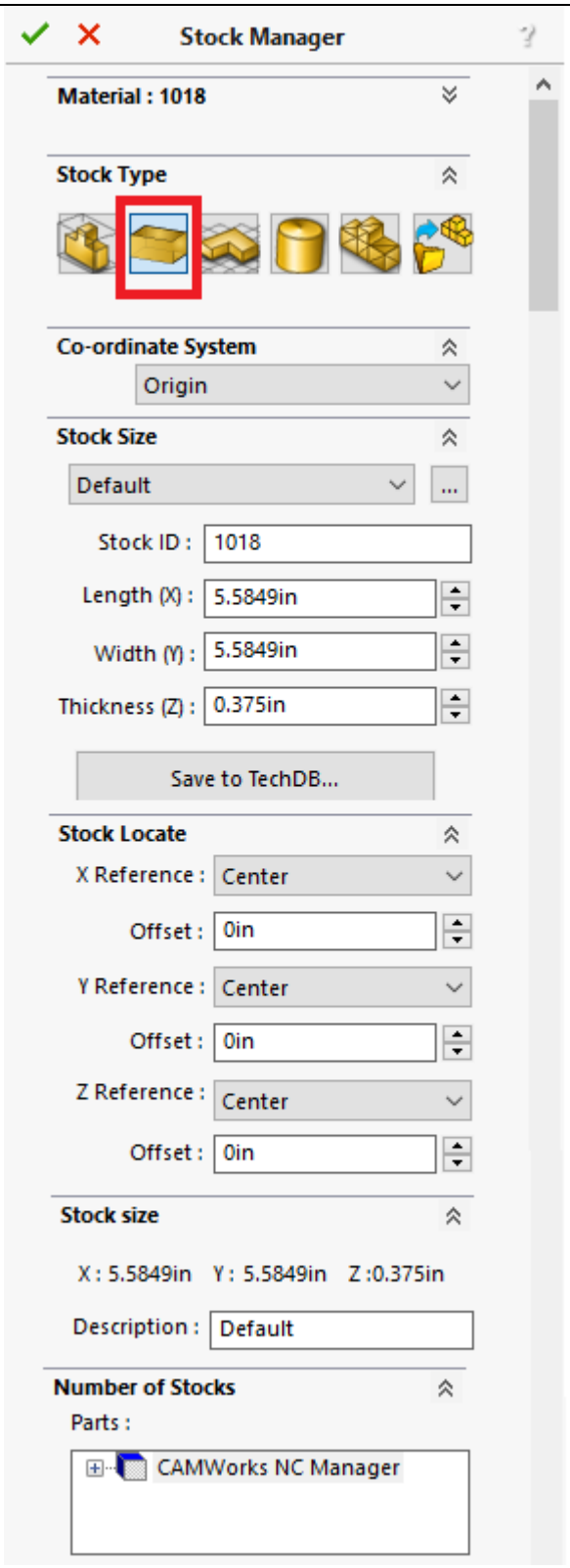
Improved - Option to Define Bounding Box Stock in Mill Mode using Pre-Defined Blocks

Purpose:

Option to define Bounding Box Stock for Mill part/assemblies using Pre-Defined Blocks

Implementation:

In previous versions of *CAMWORKS*, the option to define the stock for a mill part model/ assembly from a Bounding Box is available. Users could customize this stock by assigning the dimensions and offsets for the stock.
From *CAMWORKS 2021* version onwards, an additional option for defining the bounding box stock with pre-machined blocks will be available. For such pre-machined blocks, the dimensions of the blocks are pre-defined. User can select one of the pre-machined bounding boxes from the available list of blocks. The stock can be easily aligned with the part using different options available within the *Stock Manager* dialog box.
These pre-defined stock sizes can be sorted based on the stock material group. Any changes to the pre-machined block can be saved into TechDB. Also, user can define the pre-machined blocks and assign them to a unique stock material group in TechDB.



Parameters Displayed within Stock Manager UI when Pre-Defined Bounding Box option is selected

New - Option to Define Peck Amount for Point to point Operations in Denomination of Tool Diameter/Flute Length

Purpose:

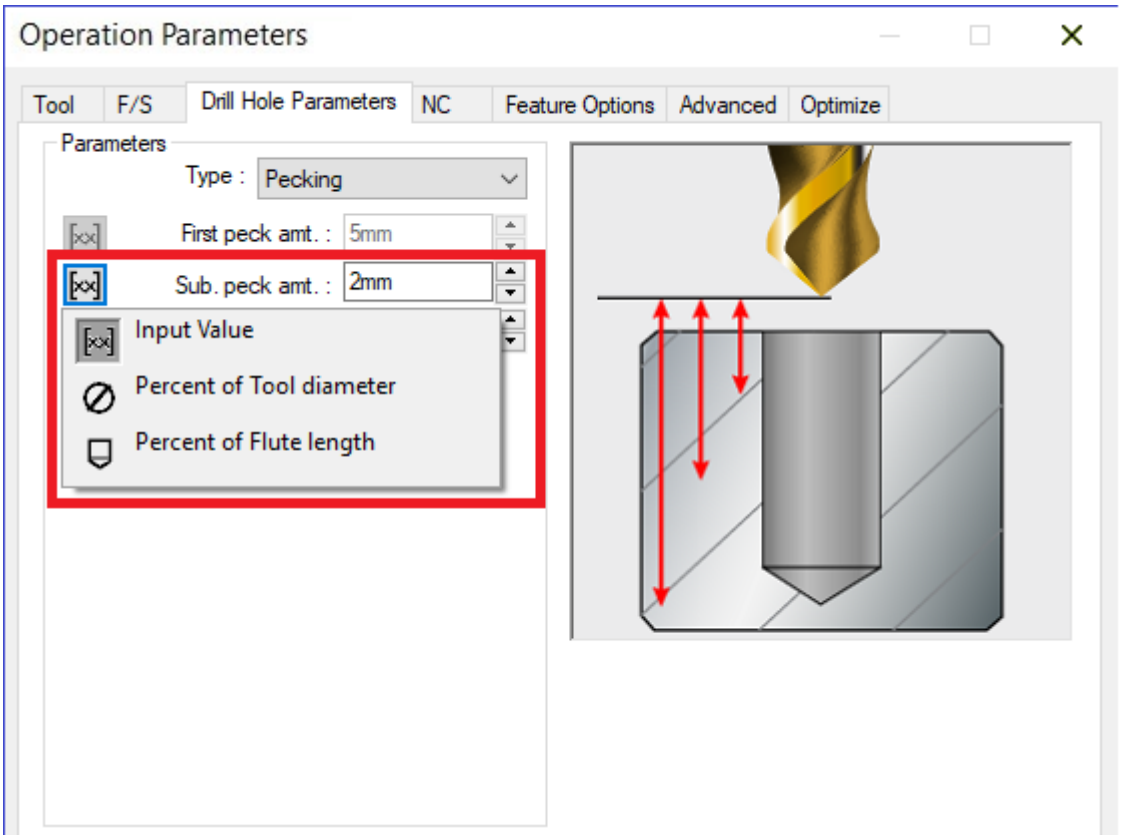
Option to Define Peck Amount for Point to point Operations in the Denomination of Tool diameter or Flute Length

Implementation:

In previous versions of ***CAMWORKS***, for Pecking cycles of Point to Point operations, the user could define the peck amounts (First peck; Sub peck amount, Minimum Peck Amount) in absolute values only. From *CAMWORKS 2021* version onwards, the option to define the peck amount with respect to the tool diameter or its flute length.

A button will be provided adjacent to each Peck amount value field in the Operation specific tab of Point to Point operations. Use this button to determine the method for assigning the Peck Amount Value. Following are the options available:

- **(Input Value)**: When this option is selected, the Peck amount will be defined using absolute values.
- **(% of Tool Diameter)**: When this option is selected, the Peck amount will be defined as a percentage of the assigned tool's diameter.
- **(% of Flute Length)**: When this option is selected, the Peck amount will be defined as a percentage of the assigned tool's Flute Length.



New button for assigning Method to Define Peck Amount in Operation Specific Tab of Operation Parameters Dialog box

Improved - Enhancements to Three Point Bore/Boss Methods for Probing Cycles

Purpose:

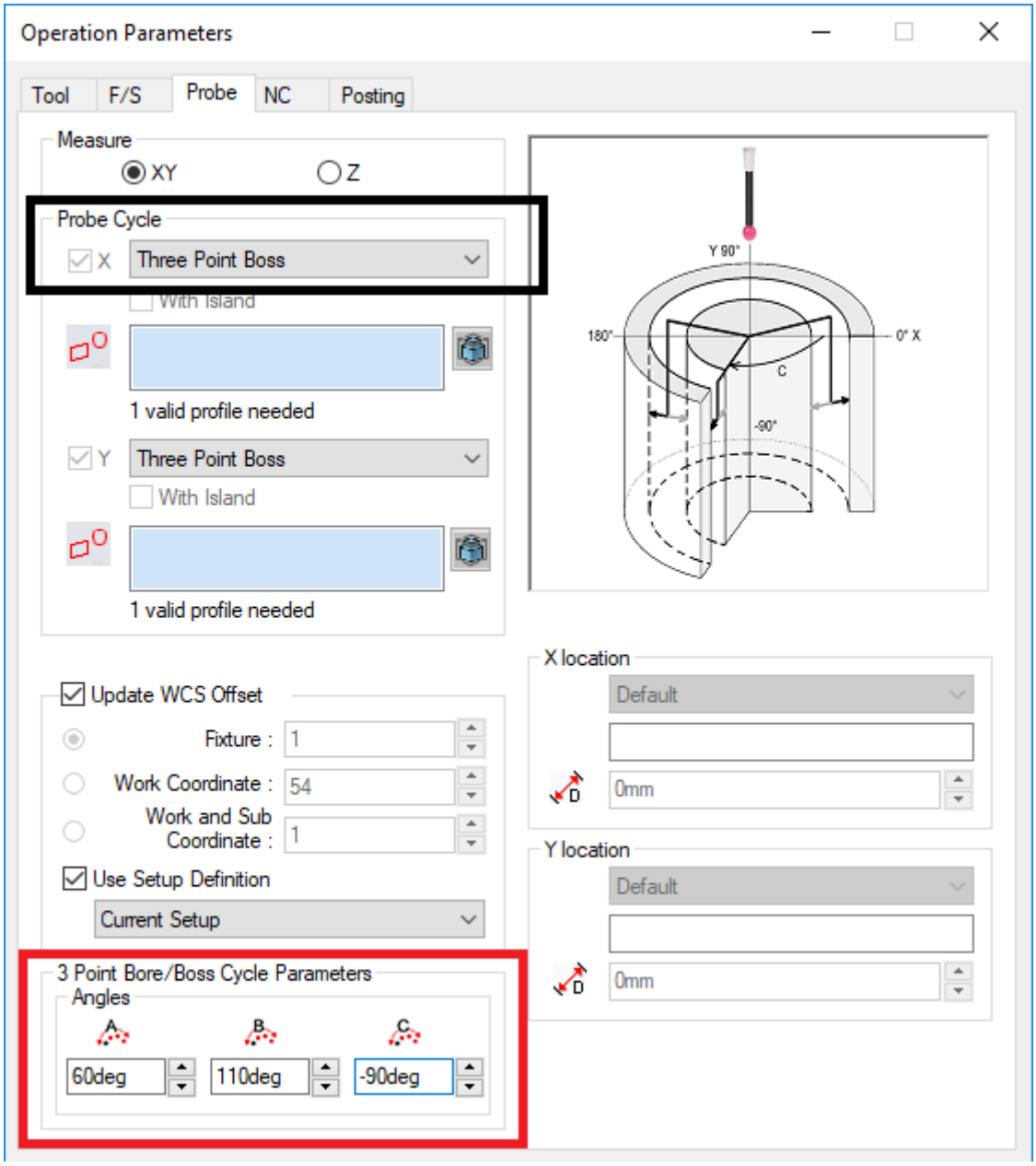
Option to modify parameters associated with *Three Point Bore* and *Three Point Boss* Probing Cycles

Implementation:

In previous versions of ***CAMWORKS***, for the *Three Point Bore* and *Three Point Boss* methods of Probing cycles, the three points probed on the feature were always equidistant (120 degrees) from each other. No option was available to modify the angular distance.

From *CAMWORKS 2021* version onwards, this option has been introduced in the form of parameters within the ***3 Point Bore/Boss Parameters*** group box under the ***Probe*** tab for a Probing operation. Parameters within this group box will be enabled only when the ***Probe Cycle*** method is set to ***Three Point Bore***, ***Three Point Boss*** or ***Automatic***.

Use the parameters in this group box to specify the angular distance between probe touch points.



'3 Point Bore/Boss Cycle Parameters' Group Box under Probe Tab

Improved - Additional Parameters for Probing Cycles

Purpose:

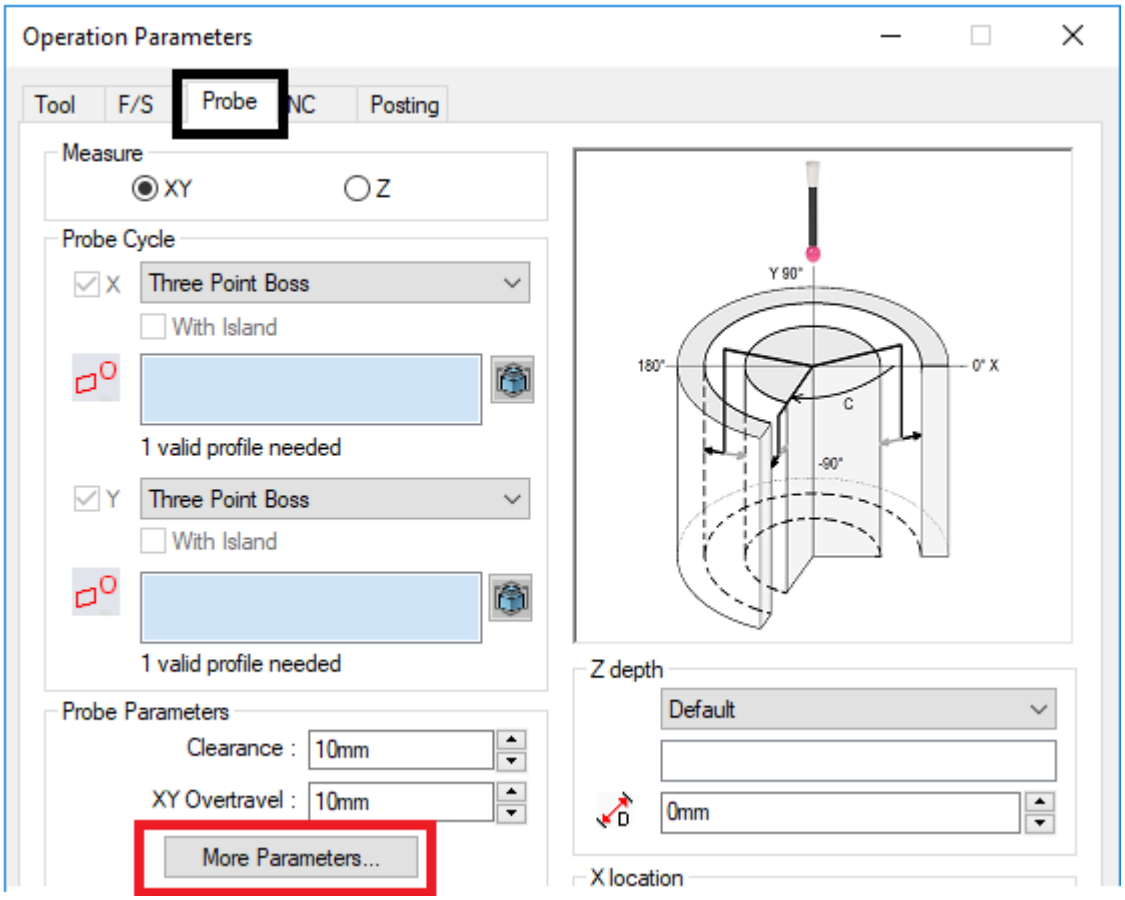
Provide user interface for Probing Cycles so that users can specify additional probing parameters that are applicable only for certain Probing Cycles or specific use cases

Implementation:

In previous versions of ***CAMWORKS***, all the Probing specific parameters were available under the ***Probing*** tab of *Operation Parameters* dialog box for Probing operations.

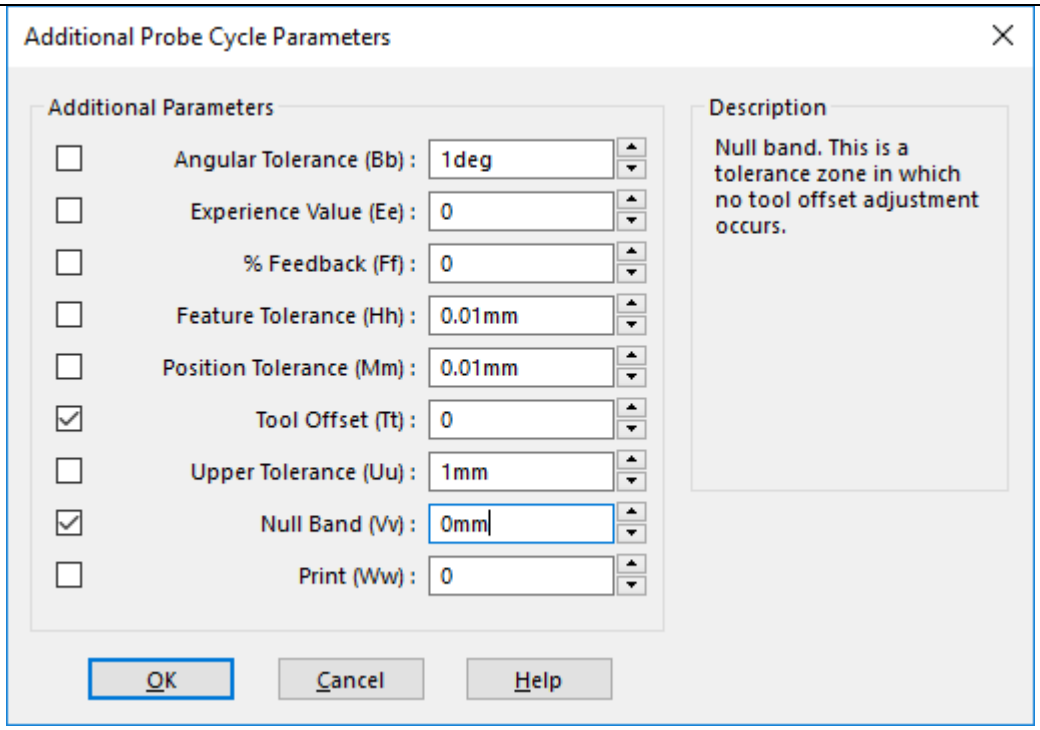
Under specific circumstances or for specific Probing cycles, it becomes necessary to specify additional parameters such as the tolerance of the feature being probed, the angular tolerance when measuring the angle between two faces, etc. Such parameters are applicable only in certain circumstances.

From *CAMWORKS 2021* version onwards, the user interface to assign values to all such additional Probe Parameters has been made available in the ***Additional Probe Cycle Parameters*** dialog box. This dialog box will be displayed when you click on the ***More Parameters*** button within ***Probe Parameters*** group box under the *Probe* tab.



'More Parameters' Button within 'Probe Parameters' group box under under Probe Tab

For all these parameters listed in the ***Additional Probe Cycle Parameters*** dialog box, you can choose whether the assigned values are to be post-processed or not. To output these values in the post-processed code, place a check in the checkbox to the left of the parameter's name. If unchecked, the value will not be post-processed.



Additonal Probe Cycle Parameters Dialog Box

New - Option to Specify whether Cutter Tool will roll over Surface Edges or be Clipped to Tool Contact Point

Purpose:

Checkbox option labelled ***Waterfall Ends*** to specify whether the toolpath will roll over surfaces edges or be clipped to the tool contact point

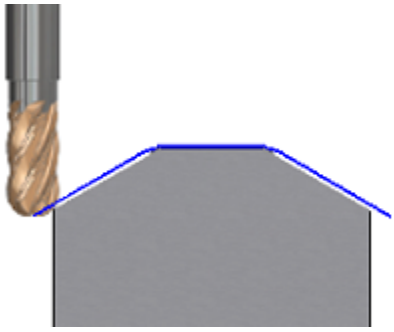
Implementation:

The ***Waterfall Ends*** checkbox option enables you to specify whether the toolpath will roll over surfaces edges or be clipped to the tool contact point. It will be enabled for only for ***Constant Stepover*** and ***Pattern Project*** operations when the *3 Axis toolpath generation method* is set to *Advanced Method* in *Update* tab of *CAMWorks Options* dialog box.

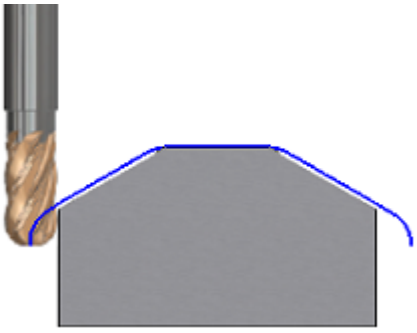
- When checked, the tool will roll over the edge of the surface a maximum distance of one tool radius.
- When unchecked, the toolpath will be clipped to the tool tangent contact point on the surface.

Note:

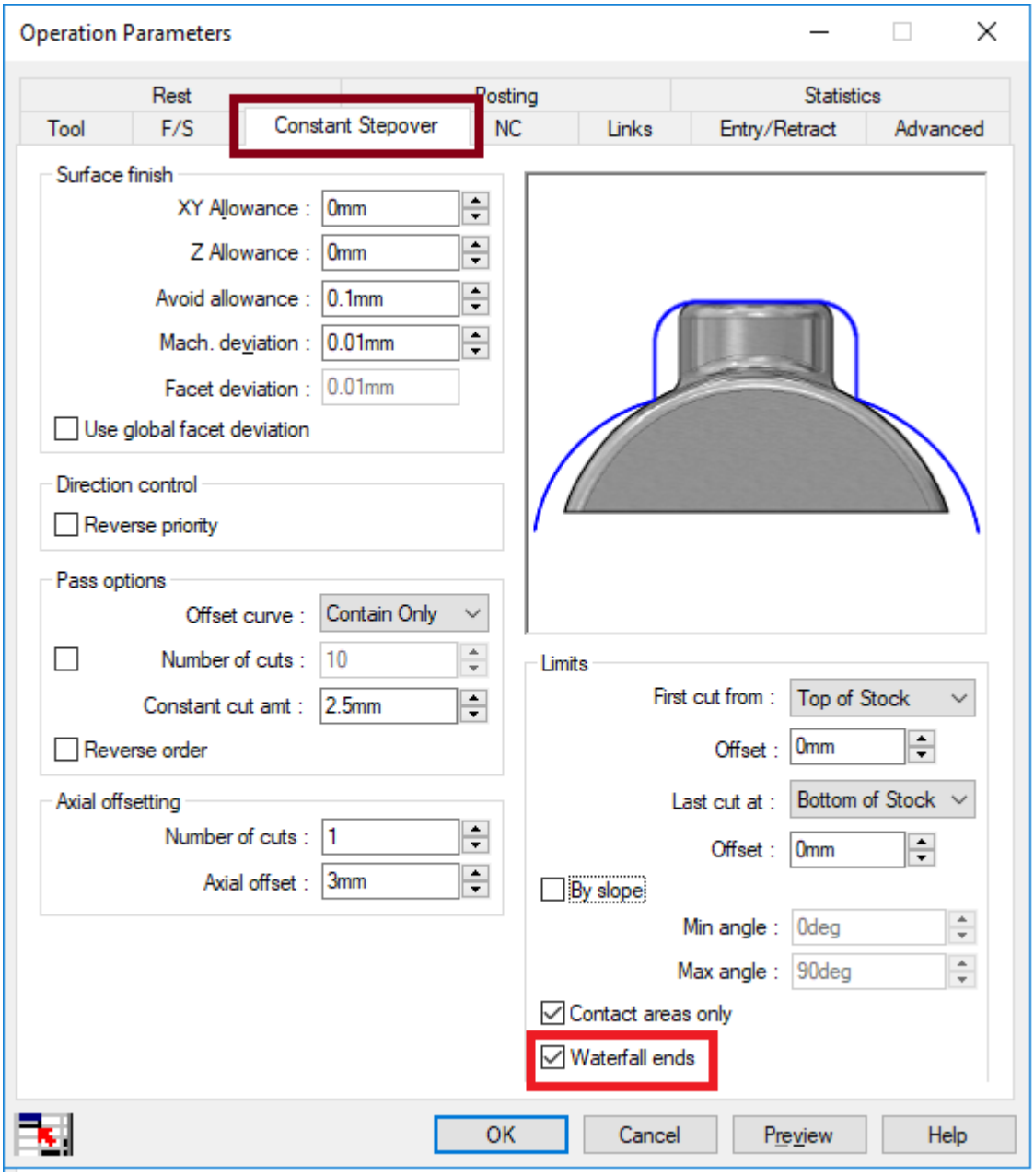
- For ***Pattern Project*** operations, this option will be enabled when the pattern in ***Pattern*** tab is set to any pattern except ***Flowline***.
- For ***Constant Stepover*** operations, this option will be enabled only if the ***By Slope*** checkbox option in ***Limits*** group box under ***Constant Stepover*** tab is unchecked.



Toolpath when "Waterfall Ends" option is unchecked



Toolpath when "Waterfall Ends" option is checked



'Waterfall Ends' Checkbox Option in Constant Stepover Tab

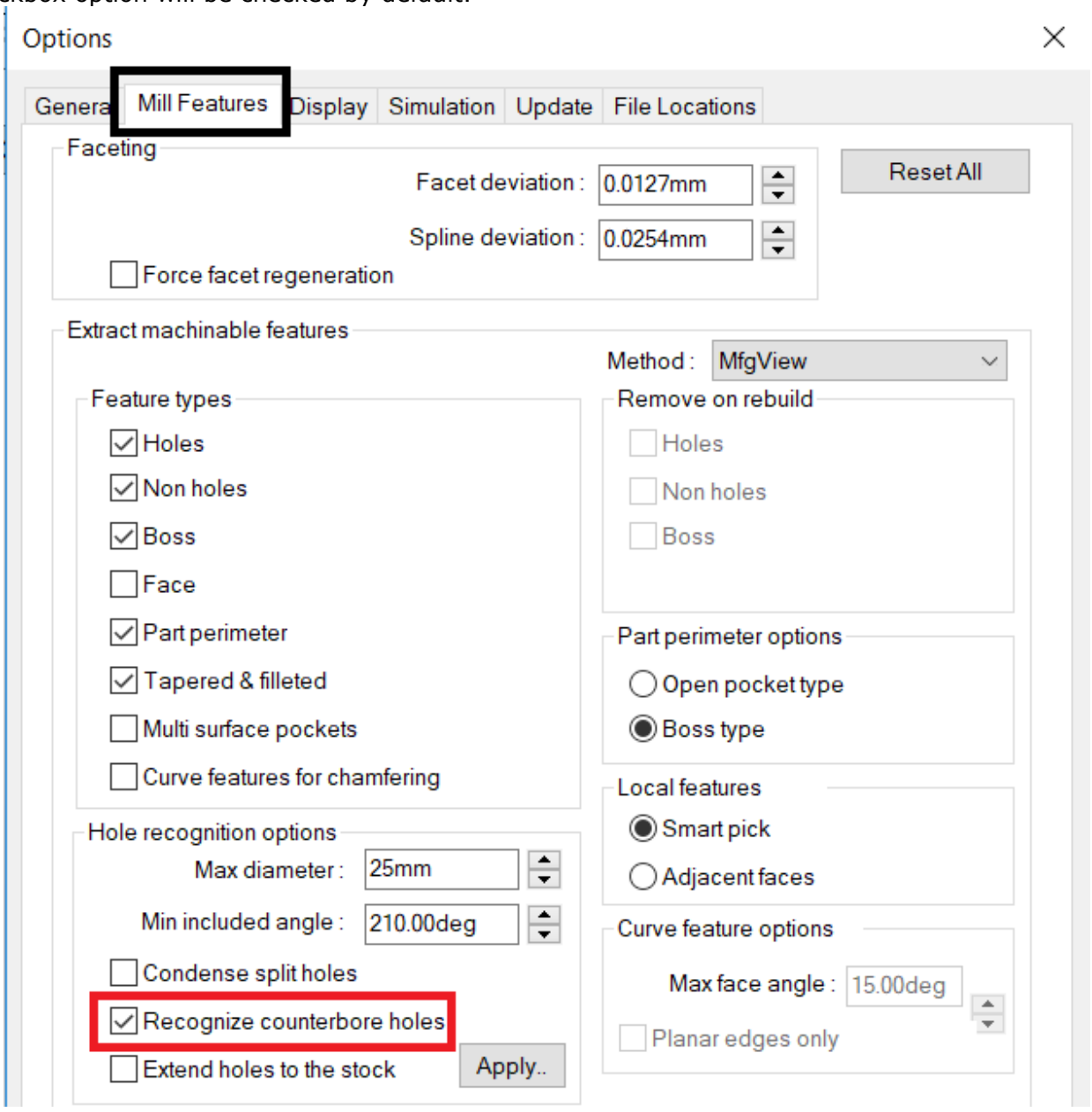
New - Option to Recognize Counterbore Hole Features as Simple Hole Features

Purpose:

Option to recognize steps of the Counterbore holes as individual hole features

Implementation:

Within the ***Hole Recognition Options*** group box under the ***Mill Features*** tab of the ***CAMWorks Options*** dialog box, a new checkbox option named ***Recognize Counterbore Holes*** has been introduced. This checkbox option will be checked by default.

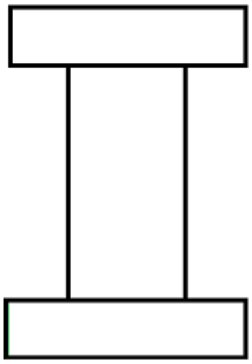
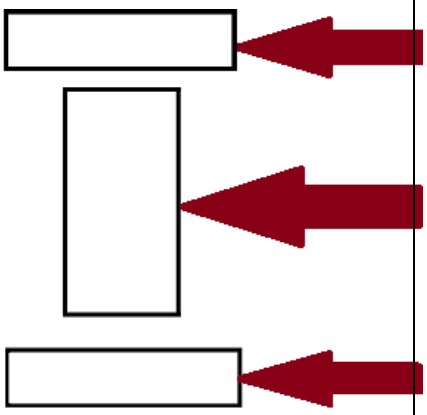
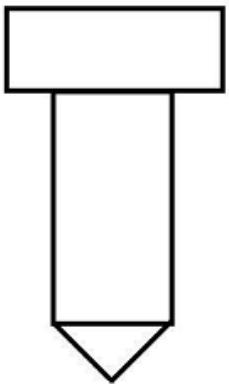
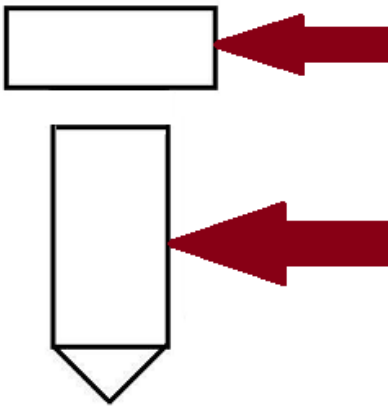


Recognize Hole Features Checkbox option in Mill Features Tab

- If this option is checked, all hole features with counterbore shape will be recognized as counterbore holes.
- If this checkbox option is unchecked, then during *Automatic Feature Recognition* of a hole feature with counterbore shape, each step will be recognized as individual simple holes. (Refer the illustration below.

Note:

If the diameter of the any of the recognized simple holes exceeds the ***Max diameter*** defined in the ***Hole Recognition Options*** group box, then it will be recognized as a circular pocket feature.

Hole Shape Recognized when "Recognize Hole Features" is checked	Hole Shape recognized when 'Recognize Hole Features" Option is unchecked	
		**Simple Hole 1** **Simple Hole 2** **Simple Hole 3**
		**Simple Hole 1** **Simple Hole 2**

New - Recognize Pockets and Slots having Non-Uniform Top or Bottom Fillets or Chamfers

Purpose:

Improvise ***Automatic Feature Recognition*** such that 2.5 Axis Mill Features having non-uniform top and/or bottom fillets, chamfers or drafts are recognized

Implementation:

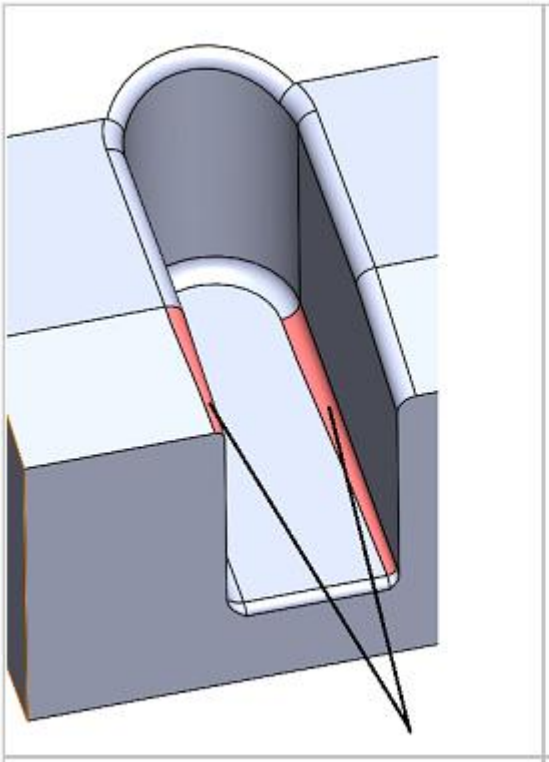
In previous versions of ***CAMWORKS***, when ***Automatic Feature Recognition*** was executed via the ***Extract Machinable Features*** command, only 2.5 Axis Mill features having uniform geometry were recognized.

Note:

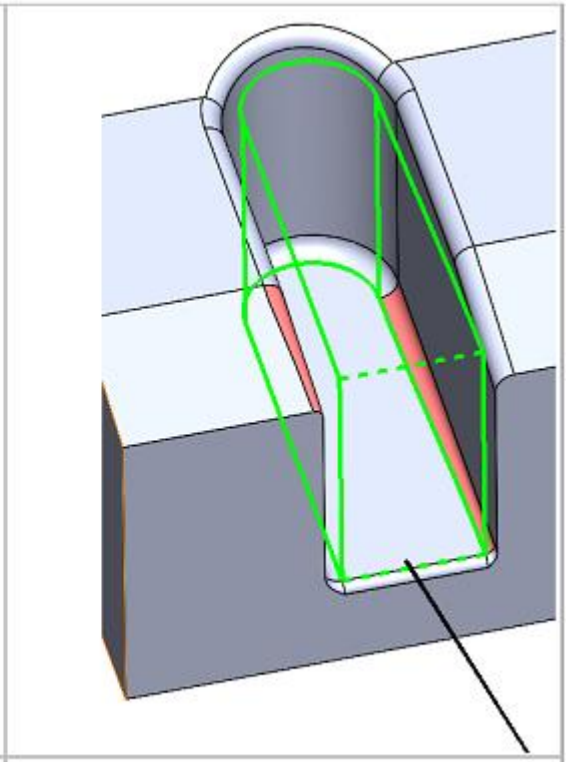
2.5 Axis Mill Tapered features will be recognized only if the Tapered and Filleted option within Features Types group box under Mill Features tab of Options dialog box is checked.

From ***CAMWorks 2021 SP0*** version onwards, the ***Automatic Feature Recognition*** functionality will be enhanced such that additional 2.5 Axis Mill pocket and slot features will be recognized in the areas of the part model having non-uniform fillets, chamfers, or drafts. Such pocket and slot features will be recognized subject to the following conditions:

i. The bottom of the pocket or slot must be flat and normal to the machining direction.
ii. Any non-uniform fillets (i.e. a fillet with variable radius and height) on the feature to be recognized will be ignored.



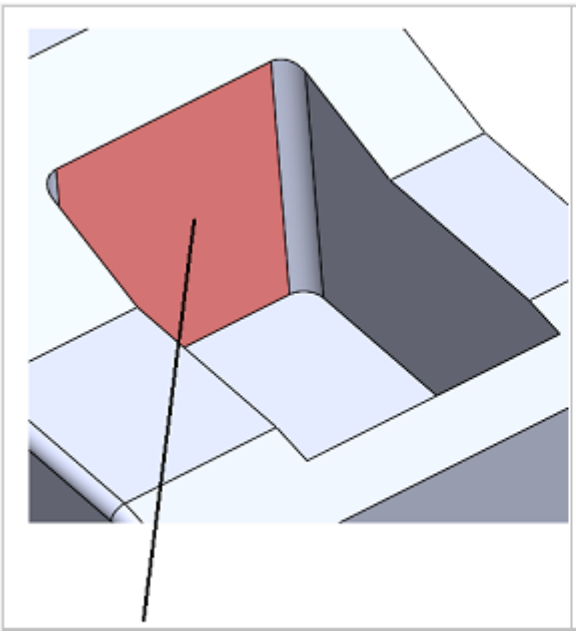
Feature with Variable Fillets



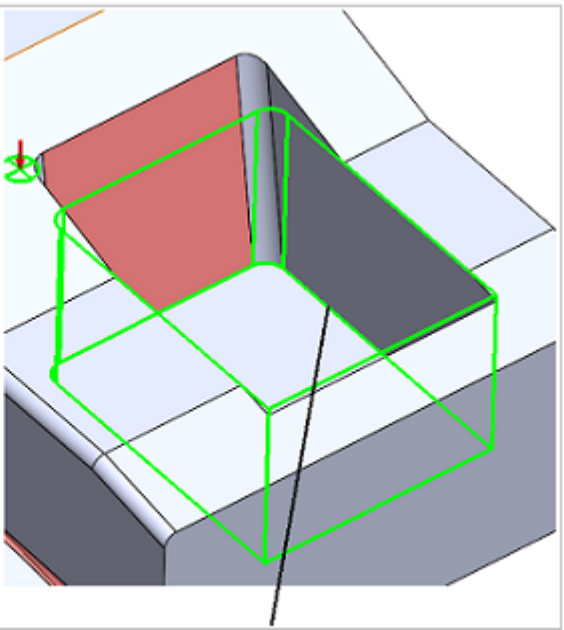
Profile of Slot Feature Recognized by Ignoring Variable Fillets

iii. Any non-uniform chamfers (i.e. a chamfer with variable angle and size) on the feature to be recognized will be ignored.

iv. If a feature on the part model has a surface with non-uniform draft, then the feature will be recognized without considering the surface with draft.



Feature that has a surface with draft and fillet



Profile of Rectangular Pocket Feature recognized by ignoring draft surface and fillet

Technology Database

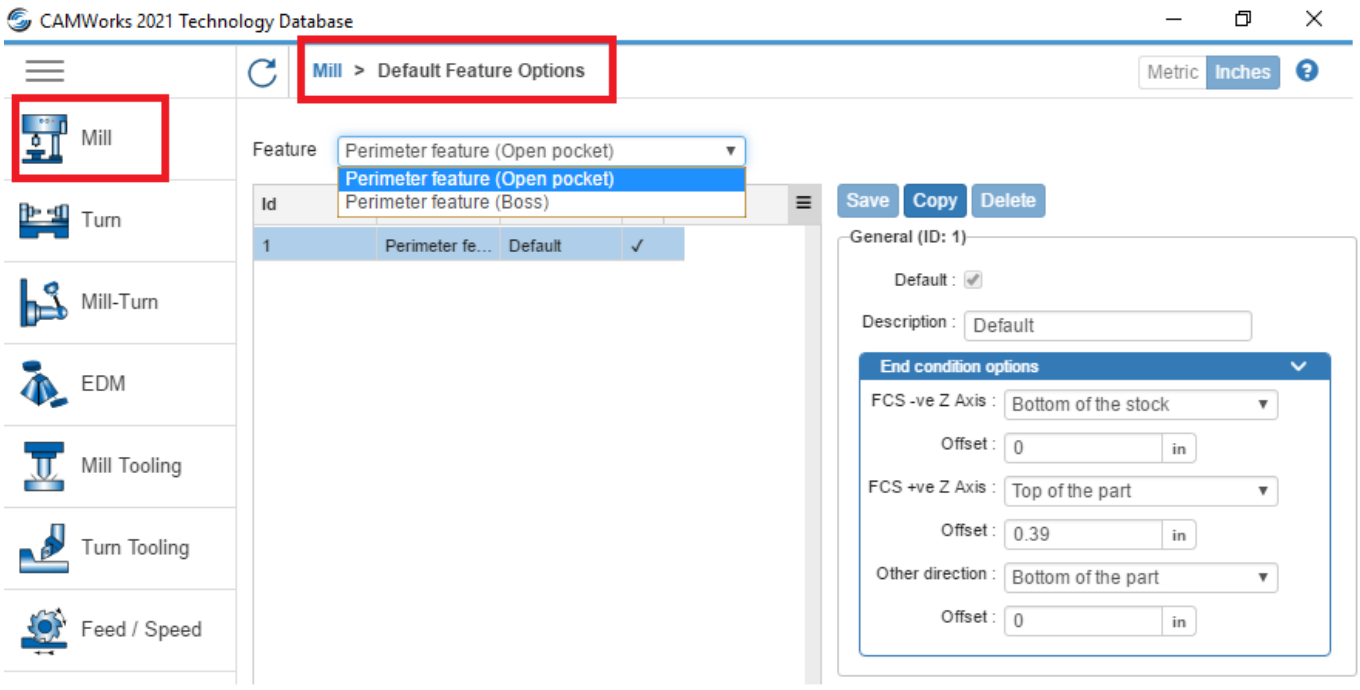
New - Assigning Default Feature Options for Part Perimeter Features in TechDB

Purpose:

Provision to assign default parametric values for Part Perimeter Features in TechDB

Implementation:

There are two feature types for the ***Part Perimeter*** feature viz. ***Perimeter feature (Open pocket)*** and ***Perimeter feature (Boss)***.
In previous versions of CAMWorks, for both these Part Perimeter feature types, there was no provision to assign different default end conditions for the different Mill Setups of the part model/ assembly.
From the ***CAMWorks 2021 SP0*** version onwards, a new user interface named ***Default Feature Options*** user interface has been introduced in TechDB under the ***Strategies*** sub-menu of ***Mill*** menu. Use this interface to define, edit, view and/or delete the End Conditions for the Part Perimeter feature types.
Multiple End Conditions can be defined but only one of them can be assigned as the default. The presence of a default end condition for Part Perimeter feature in TechDB further automates the insertion of Part Perimeter feature within the CAMWorks user interface.
Within CAMWorks, when inserting a new Part Perimeter feature via ***Automatic Feature Recognition*** (AFR) or ***Interactive Feature Recognition (IFR)***, the default End Conditions for the part perimeter feature being inserted will be determined by the default values assigned within this user interface of TechDB.



'Default Feature Options' User Interface in Mill Menu of Technology Database

New - Option to Re-sequence Operations in the Features & Operations User Interface

Purpose:

To provide an option to re-sequence operations listed in the Operations table for ***Feature and Operations*** user interface and other user interfaces where operation sequences have been defined

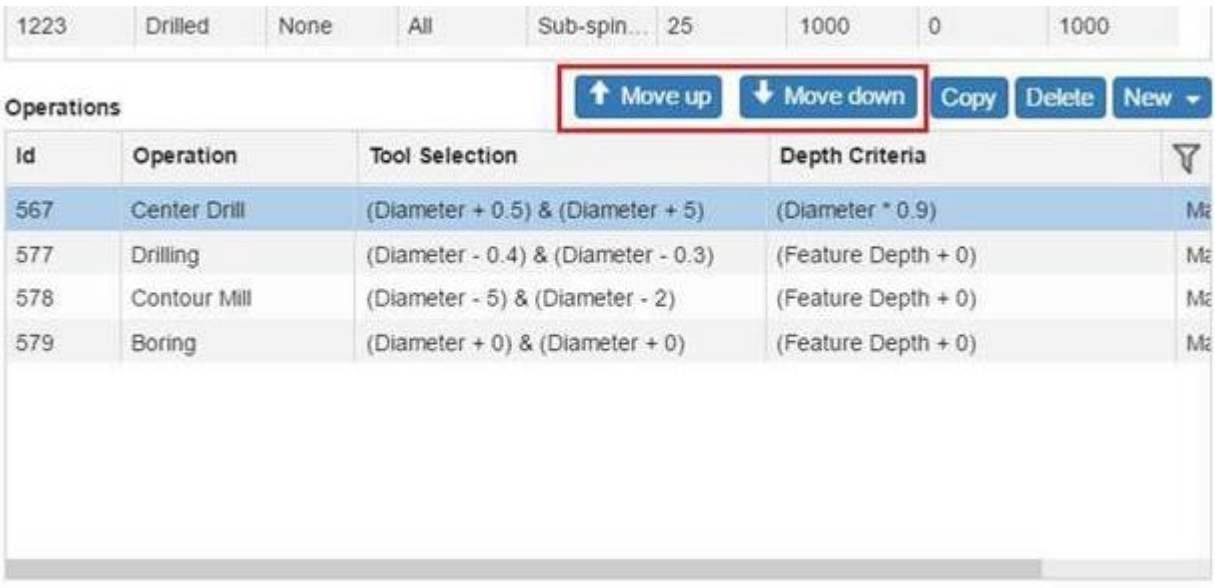
Implementation:

Within the *Technology Database* application, the ***Feature and Operations*** user interfaces for Mill, Turn and EDM and other user interfaces (such as Multi-stepped Holes, Thread Mill, Tap Cutting, Tap Rolling) are used to define/edit operation sequences that will be used to machine the different machinable features. You can create and store machining sequences of repetitive operations.

For a selected entry within these user interfaces, the corresponding operation sequence will be displayed in the ***Operations*** table.
If no operation sequence has been defined for a Feature Condition, then this table will be empty.
The commands available for the ***Operations*** table are to insert operations and create an operation sequence.
In previous versions of CAMWorks, there were no options available within this user interface to re-sequence the operations listed in the ***Operations*** table. Consequently, users could neither insert any operation between two listed operations nor change the sequence.
From ***CAMWorks 2021*** version onwards, the ***Move Up*** and ***Move Down*** command buttons have been made available in the ***Feature and Operations*** user interfaces. If two or more operations are listed in the ***Operations*** table, you can use the ***Move Up*** and/or ***Move Down*** command buttons to rearrange the operation sequence.

In Technology Database, the ***Move Up*** and ***Move Down*** command buttons will be available in the following user interfaces for the resequencing operations.

- Mill>>Strategies>>Features & Operations
- Mill>>Strategies>>Thread Mill
- Mill>>Strategies>>Multi-stepped Holes
- Turn>>Strategies>>Features & Operations
- Turn>> Strategies>>Thread Condition Operations
- EDM>> Strategies>>Features & Operations
- Mill Tooling>>Threading Strategies>>Tap – Cutting Strategies
- Mill Tooling>> Threading Strategies>>Tap – Cutting Strategies
- Turn Tooling>>Threading Strategies>>Tap – Cutting Strategies
- Turn Tooling>> Threading Strategies>>Tap – Cutting Strategies



Move Up and Move Down Buttons in Feature & Operations User Interfaces

New - Functionality to Import Tools from Harvey & Helical Tools Catalog into TechDB

Purpose:

Option to import Mill Tools from Harvey and Helical Tools Catalog into TechDB

Implementation:

From *CAMWorks 2021* version onwards, the tool catalogs of ***Harvey*** and ***Helical*** Tools (both owned by *Harvey Performance Company*) will be shipped along with CAMWorks. After CAMWorks is installed, these tool catalogs will be available in JSON (JavaScript Object Notation) format at the following location:

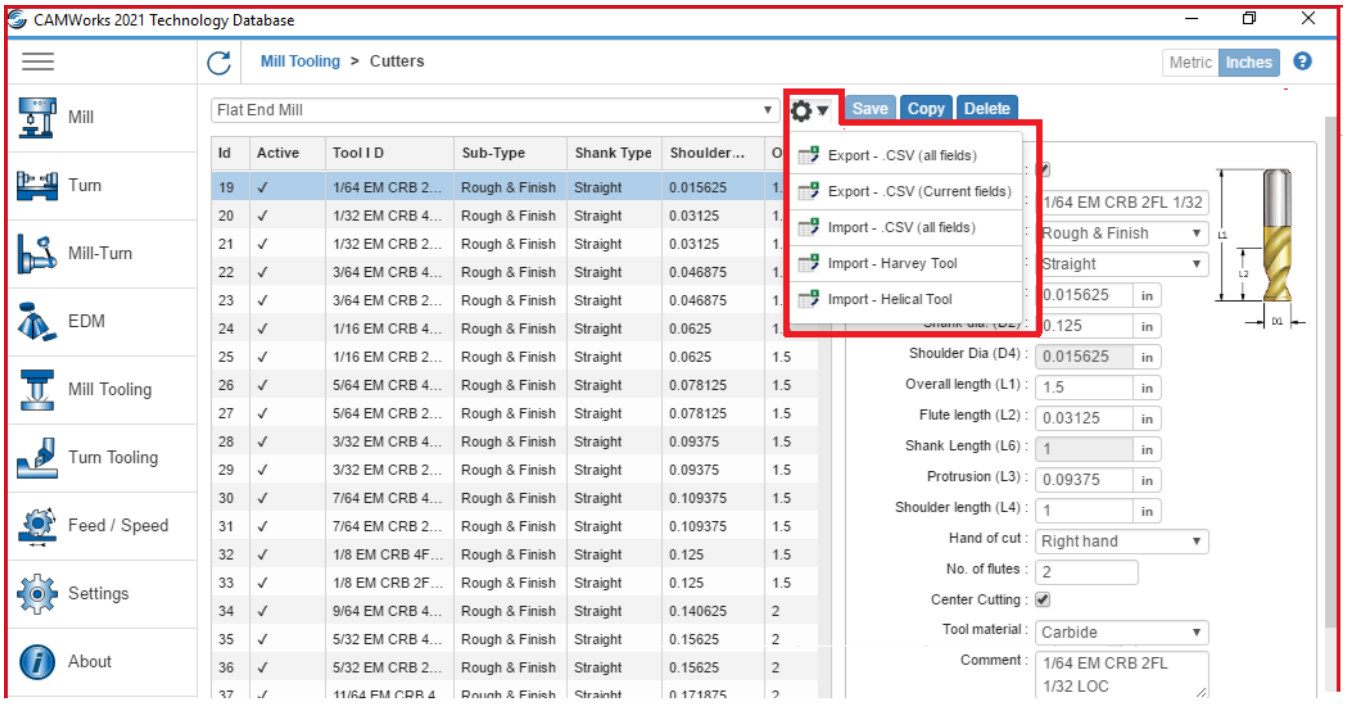
C:\CAMWorksData\CAMWorks202Xx64\Tooling\Harvey_Helical

The JSON format is an open standard file format as well as a data interchange format. It therefore allows transmission of data objects consisting of attribute–value pairs, array data types and serializable values. The tool catalogs of Harvey and Helical Tools (in JSON format) primarily contain End Mill and Taper End Mill tools. These tools can be imported into TechDB using the corresponding user interfaces for these tools.

These following command options to import Harvey and Helical tools will be available under the ***Options for Importing/Exporting Info Displayed in Tables*** menu in the user interfaces for End Mill Tools and Taper End Mill Tools in the TechDB.

- Import - Harvey Tool
- Import - Helical Tool

Executing either of these commands displays the ***Import Tools*** window. Use the parameters and options given within this window to import the desired Harvey/Helical tools into TechDB.



Option to Import Harvey & Helical Tools in End Mill & Taper End Mill User Interface of TechDB

Tools Types from Harvey & Helical Tools Catalogs can be Imported into TechDB

Tools from the tool catalogs of Harvey and Helical Tools (in JSON format) can be imported into TechDB for the following Mill Tools using corresponding user interfaces:

- Flat End Mill Tools
- Ball Nose Mill Tools
- Hog Nose Mill Tools
- Taper Flat End Mill Tools
- Taper Ball Nose Mill Tools

New - Option to Modify Post Processor Path in Technology Database

Purpose:

To provide an option within the TechDB app to the user to set the path of a folder to select the post processors.

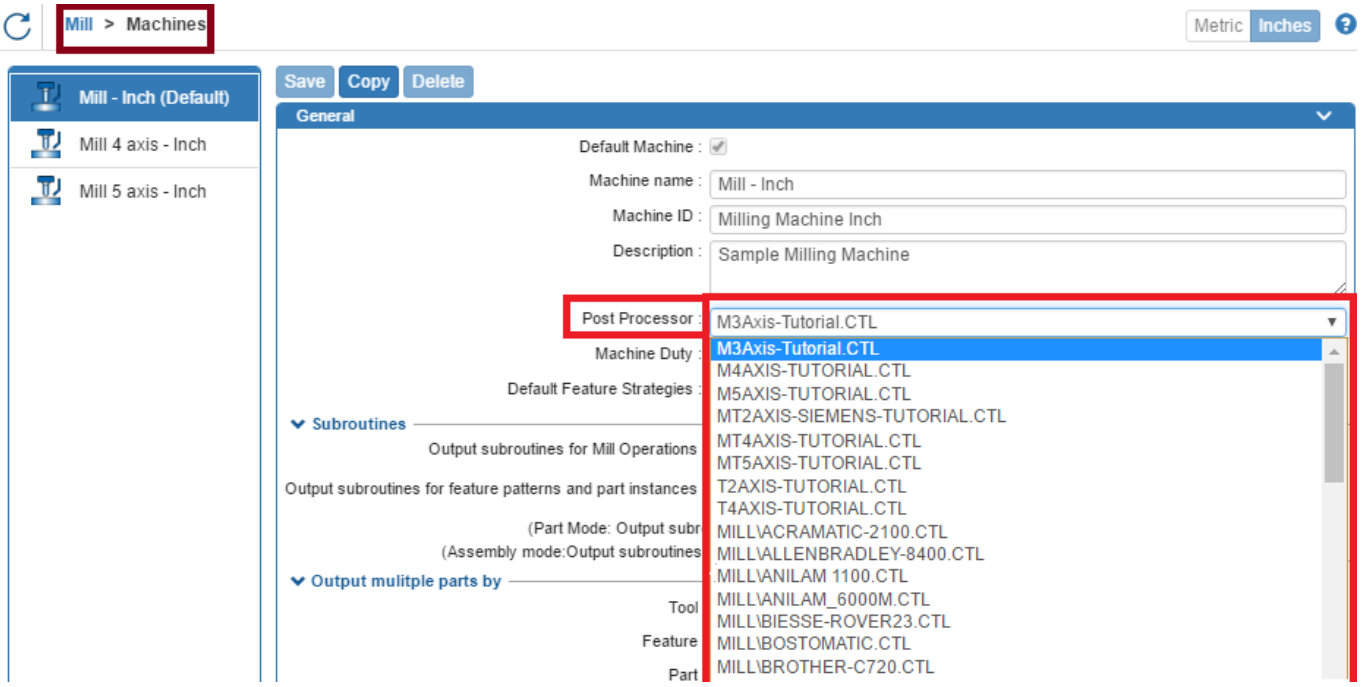
Implementation:

How the Assignment of Post Processor to a Machine is done in Previous CAMWORKS Versions

Within the *CAMWORKS* application, users can assign the Post Processor to be associated with a Mill or Turn machine in the ***Post Processor*** tab of the ***Machine*** dialog box. Using the Browse button within this tab, users can set a folder path to a specific folder. All post processors within the folder and sub folders thus specified will become available for selection in the ***Available*** list box within the *Post Processor* tab. To select the desired post processor, the user can highlight it in the ***Available*** list box and click the ***Select*** button.

Within the TechDB linked to the *CAMWORKS* application, the option to assign/reassign the post processor associated with the active machine is available in the General tab of that active Mill Machine/ Turn Machine. This option is provided in the form of the Post Processor dropdown list. It will list the same post processors present within the folder specified in the *Post Processor* tab of *Machine* dialog box in *CAMWORKS* application. (The listed post processors also include those within the sub-folders of the specified folder.)

The current limitation in previous *CAMWORKS* versions is that users cannot specify any alternate folder in TechDB from where post processors can be made available for selection when defining or editing Mill or Turn machines. The selection can be made strictly only from the folder defined in *CAMWORKS* application.

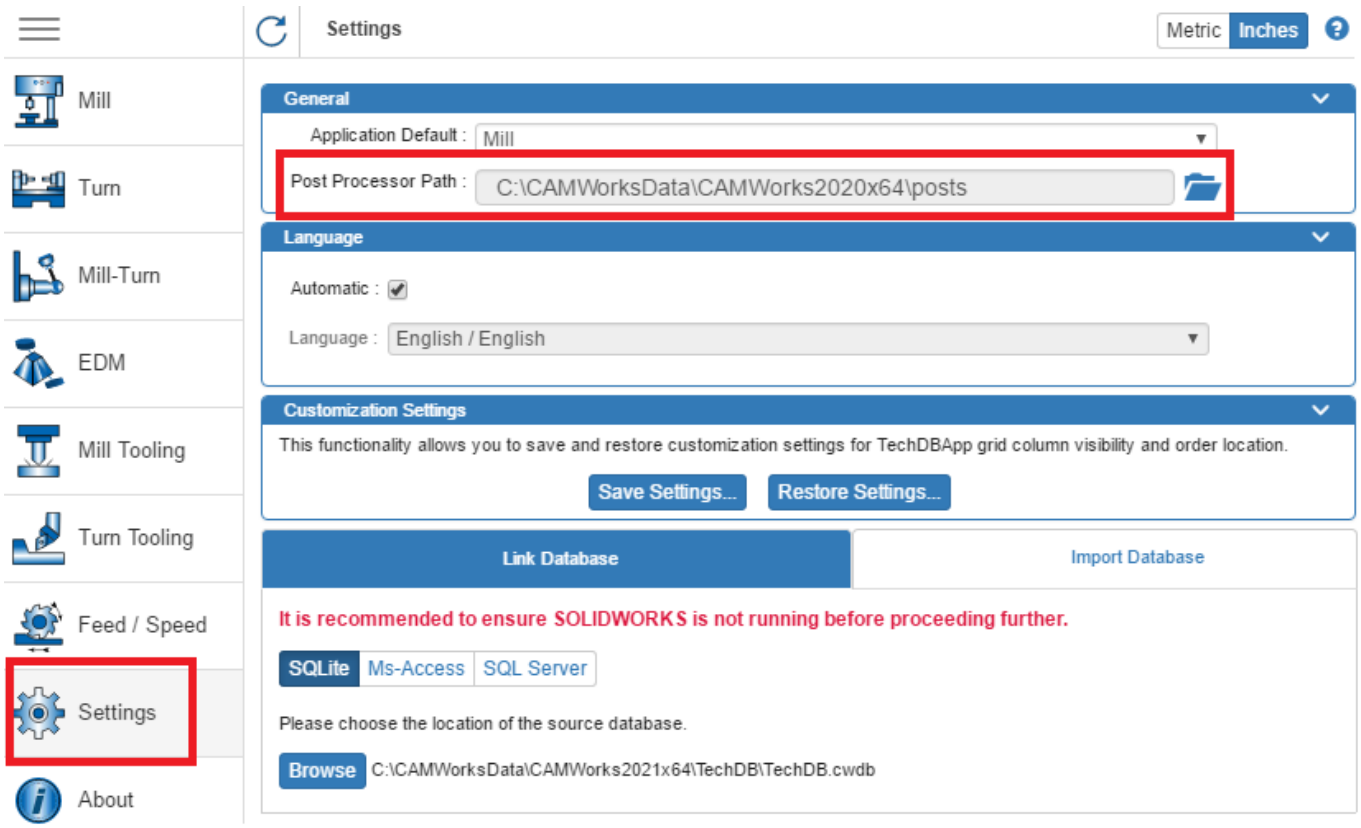


Post Processor Dropdown List in Mill Machines UI of the Technology Database App

Introduction of Option to Set Post Processor Path in SOLIDWORKS 2021 version

From *SOLIDWORKS 2021* version onwards, the option to set the folder path for post processor selection has been introduced in the ***Settings tab*** of the Technology Database UI. This is provided in the form of the Post Processor path field and associated Browse button. Under default settings, this path will be identical to the folder path specific in the *Post Processor* tab of *Machine* dialog box in *CAMWORKS* application. To change the path, click on the ***Browse*** button and use the ***Browse for Folder*** dialog box that will be displayed to select the desired folder location Once the new folder location is selected, the ***Post processor path*** field will be updated to display the new folder path. In

the ***General*** tab of the ***Mill Machine*** and ***Turn Machine*** interface within the TechDB, the ***Post Processor*** dropdown list will now display post processors present within the currently selected folder and its sub-folders.



Post Processor Path and Browse button in Settings UI of the Technology Database App

Mill-Turn

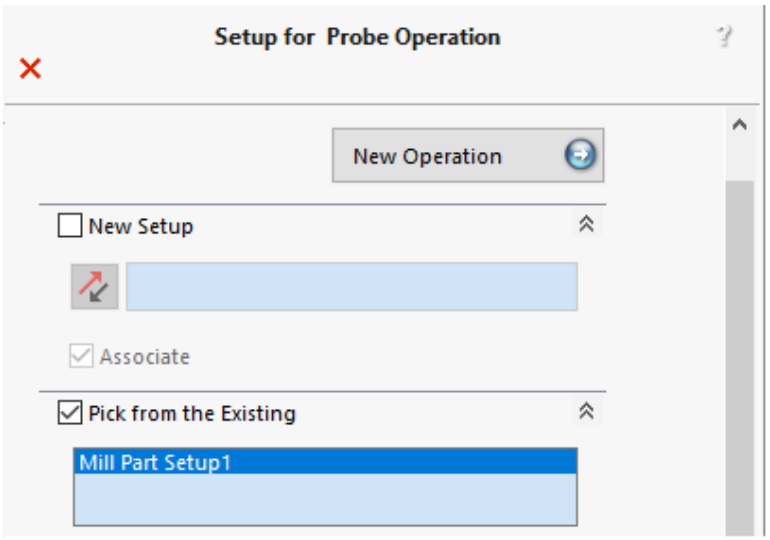
New - Support for Probe Tools and Probing Operations extended to Mill-Turn Mode

Purpose:

To extend the functionality of Probing Operations using Probe Tools available in Mill mode to Mill-Turn mode

Implementation:

Probing is an established best practice for maximizing the efficiency, quality, capability, and accuracy of machine tools.
This functionality was introduced in the *CAMWorks 2020 SP0* version for Mill mode.
From *CAMWorks 2021 SP0* version onwards, this functionality has been extended to the Mill-Turn mode too. Probing operations can be added under Mill Part Setups in the Mill-Turn mode. The user interface related to the Probing operations will be identical to those available in the Mill mode.



Setup For Probe Operation Dialog box

Multiaxis Mill

Improved - Additional Parameters for Multiaxis Milling Operations

Purpose:

Provide additional parameters for Multiaxis Milling operations to ensure additional flexibility when defining those operations

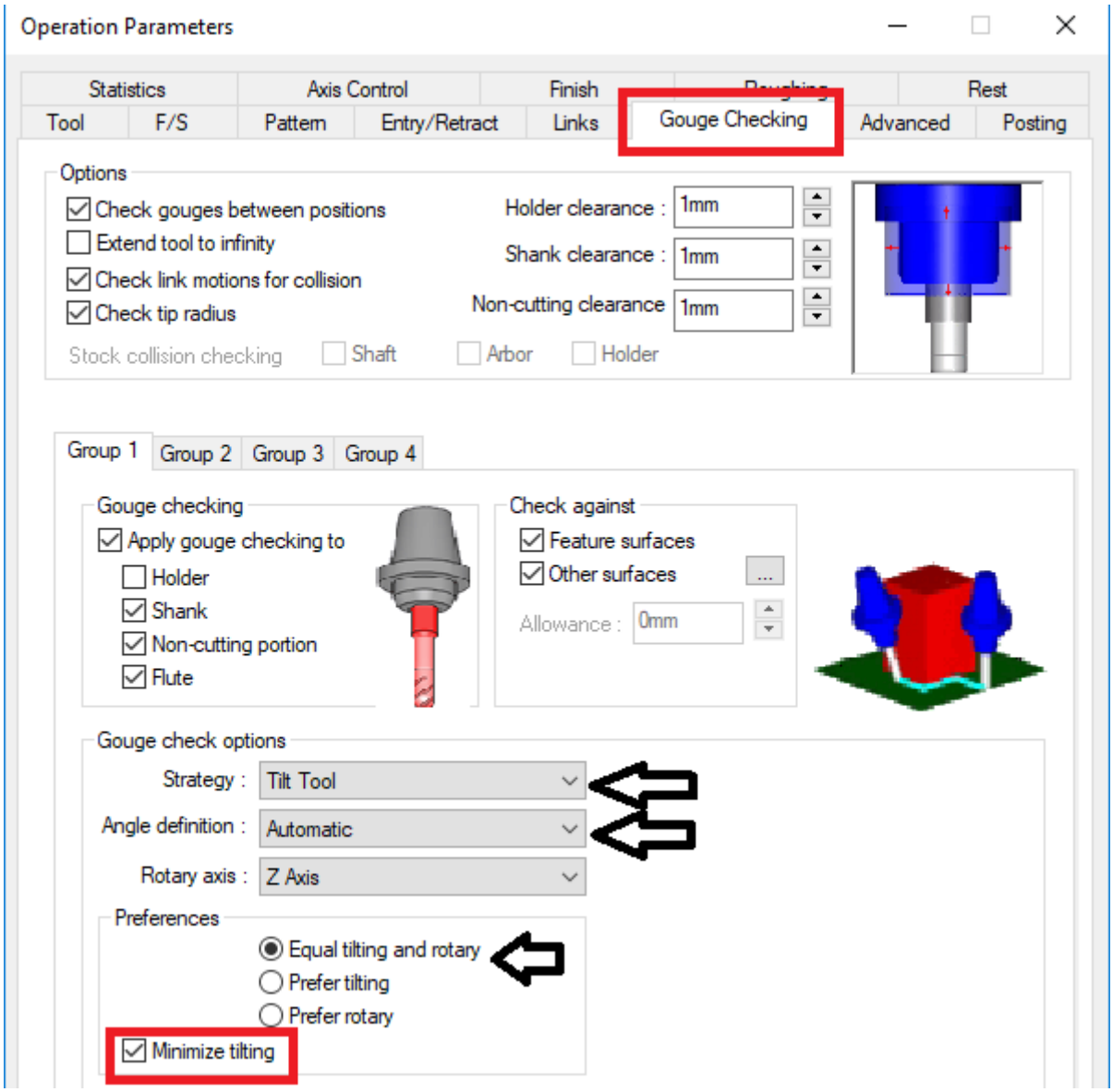
Implementation:

From *CAMWorks 2021* version onwards, the following additional parameters will be available in the various tabs of the Multiaxis Mill operations:

Gouge Checking Tab for Multiaxis Milling

Within this tab, new checkbox option for minimizing titling will be available. This checkbox will be enabled when the following conditions are fulfilled:

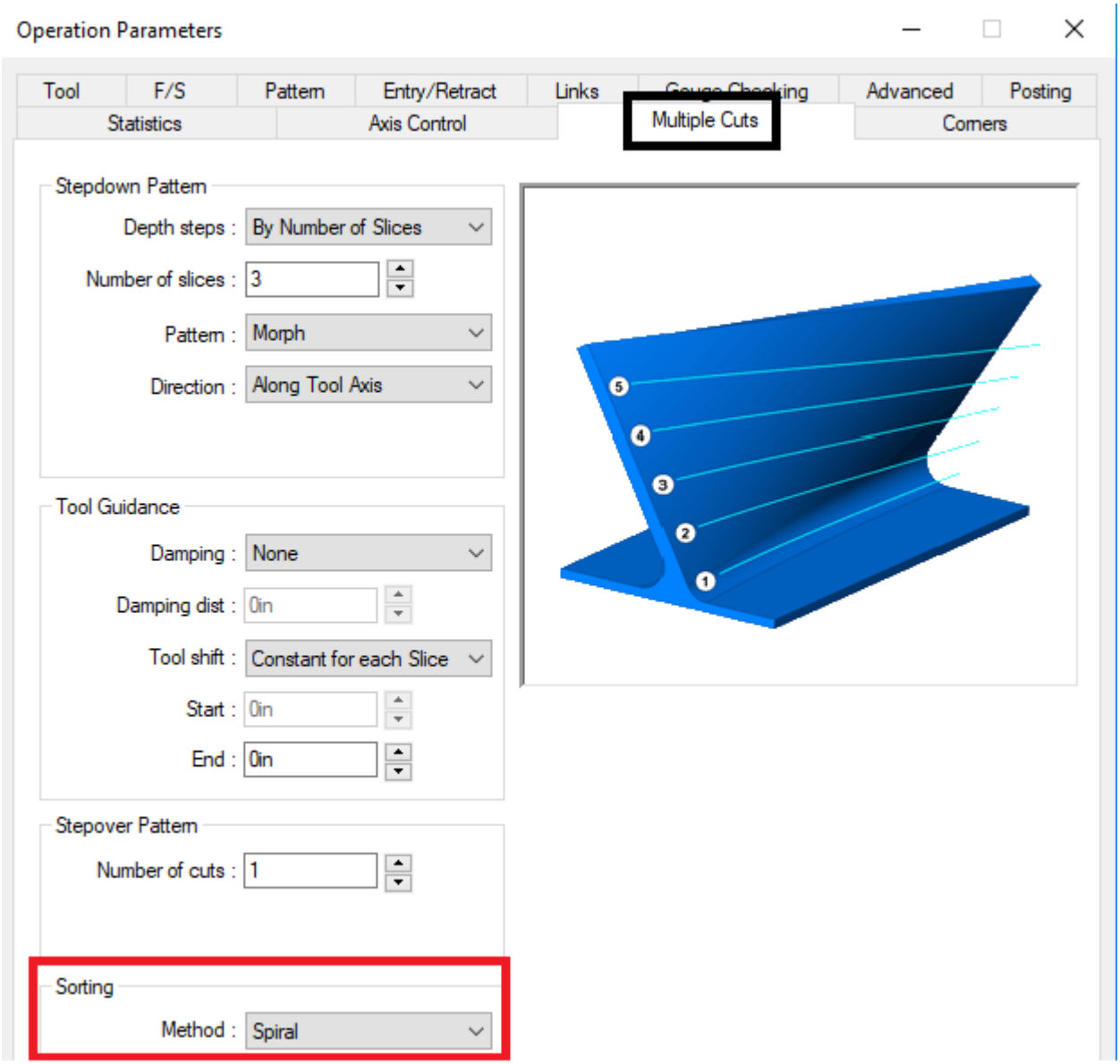
- Gouge Checking Strategy is set to ***Tilt tool***
- Angle Definition is set to ***Automatic***
- The option selected in the ***Preferences*** group box is set to ***Equal tilting and rotary***



Minimize tilting option introduced in Gouge Checking tab

Multiple Cuts Tab for Multiaxis Swarf Milling

Within this tab, the new Sorting Method of Spiral will be available. When this option is selected, CAMWorks will apply a spiral method to the stepdown and stepover passes. This method is only available for Stepdown when ***Pattern*** is set to ***Morph*** and ***Direction*** option of ***Along Tool Surface*** is not used.

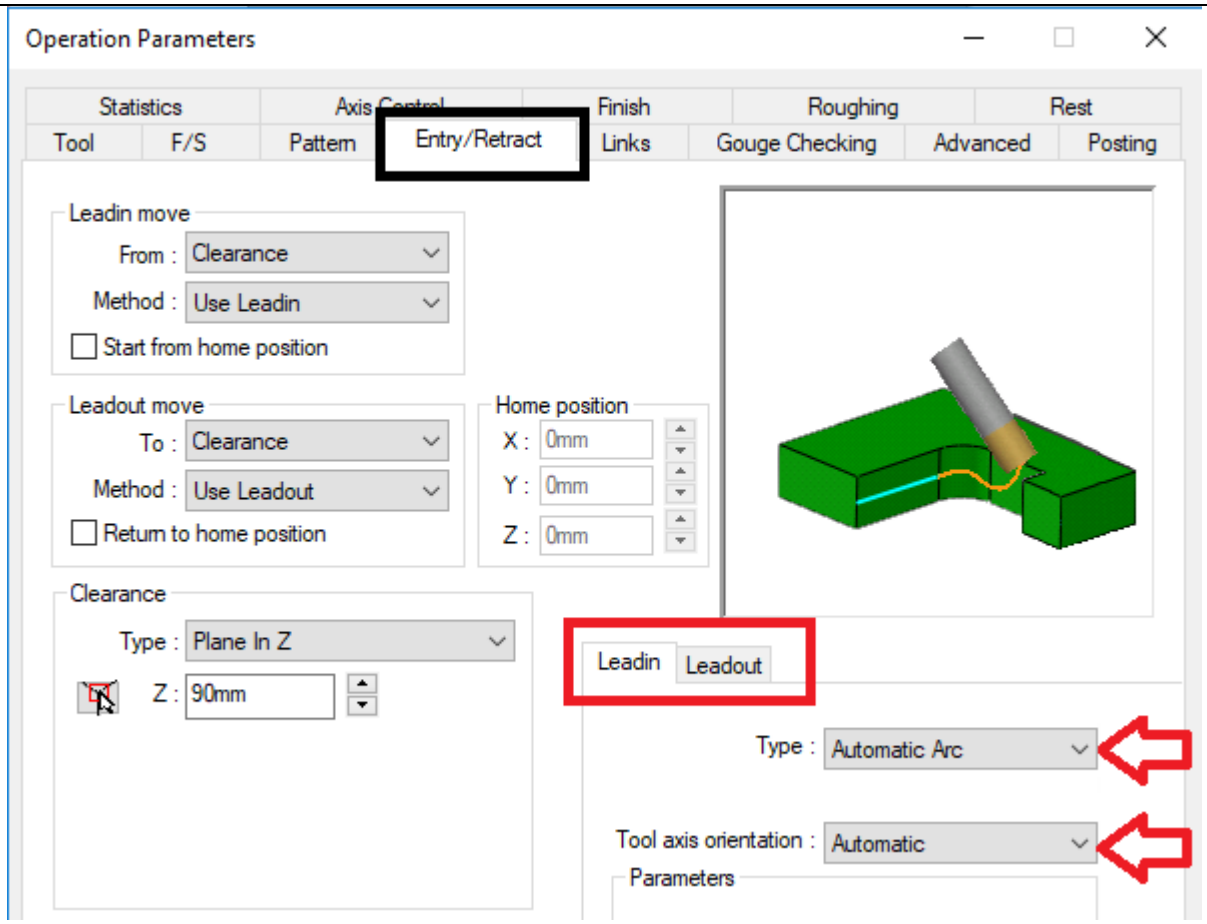


'Spiral' Method for Sorting in Multiple Cuts Tab for Multiaxis Swarf Milling Operation

Entry/Retract Tab for Multiaxis Swarf Milling

Within this tab, the ***Tool Axis Orientation*** dropdown list parameter will be enabled in the Leadin and Leadout pages when Type is set to Automatic Arc. Additionally, a new option ***Automatic*** will be available in the ***Tool Axis Orientation*** dropdown list.

When the ***Automatic*** option is selected in the ***Tool Axis Orientation*** dropdown list, CAMWorks will automatically determine the Tool orientation along the leadin/leadout arc. This option is only available when the leadin/leadout Type is set to ***Automatic Arc***.



New option 'Automatic' in Tool Axis Orientation Dropdown List in Leadin/Leadout Pages under Entry/Retract tab

Turn/ Mill-Turn

New - Option to retain Arc Moves for B Axis Continuous Turn Toolpaths

Purpose:

To provide the option to users to retain arc moves for B Axis Continuous Turning Toolpaths

Implementation:

In previous versions of CAMWorks, when Continuous B Axis Turning toolpaths were generated, any arc moves present were converted to line moves.

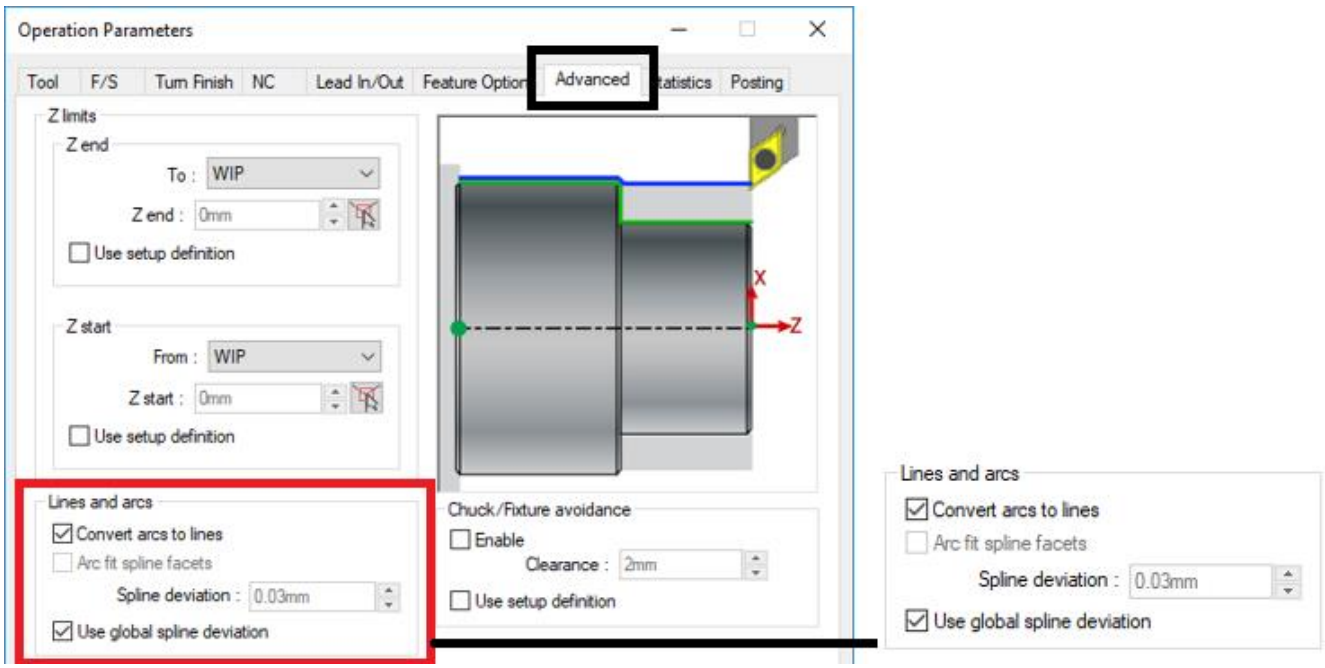
From ***CAMWorks 2021 SP0*** version onwards, the option to retain or not retain arc moves will be provided in the ***Advanced*** tab for ***Turn Finish*** operations in the form of the ***Convert arcs to lines*** checkbox option.

The group box named ***Spline Output*** present under ***Advanced*** tab for Turn operations has been renamed to ***Lines and Arcs***. The newly introduced ***Convert arcs to lines*** checkbox option will be available within this group box.

How the 'Convert arcs to lines' Checkbox Option will work

This checkbox option will be enabled only when the ***Continuous B Axis*** checkbox within ***Continuous B Axis Turning*** group box under ***Turn Finish*** tab for a ***Turn Finish*** operation is checked. This checkbox will be checked by default. It will be disabled for Turn operations other than *Turn Finish*.

- When unchecked, arc moves generated for the *Continuous B Axis Turn* toolpath will be retained and not converted to linear moves.
- When checked, arc moves generated for the *Continuous B Axis Turn* toolpath will be converted to linear moves.



'Lines and arcs' group box with 'Convert arcs to lines' checkbox option under Advanced tab for Turn Finish Operations

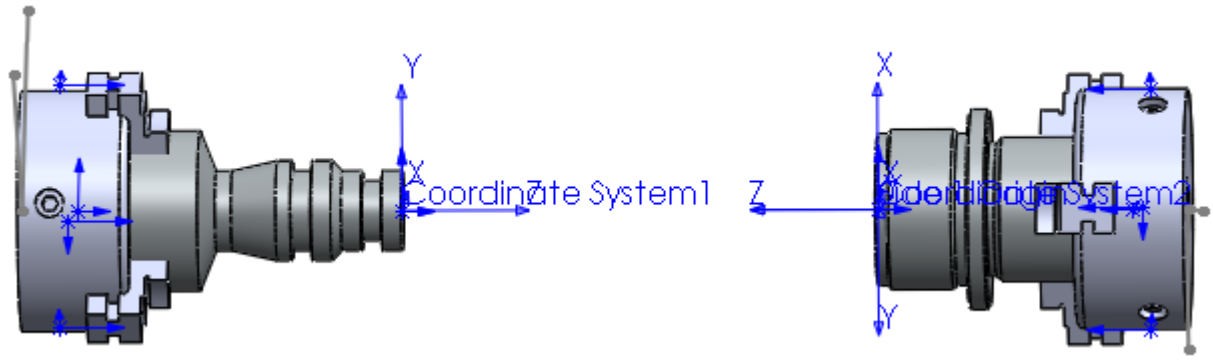
New - Assembly Mode Support for Turn and Mill-Turn

Purpose:

To extend the Assembly support function of Mill mode to Turn and Mill-Turn modes of CAMWorks

Implementation:

From *CAMWorks 2020 SP3* version onwards, Assembly mode functionality will be available for Turn and Mill-Turn mode of CAMWorks. In this Turn/Mill-Turn Assembly mode, assemblies containing a max of two-part models will be supported.

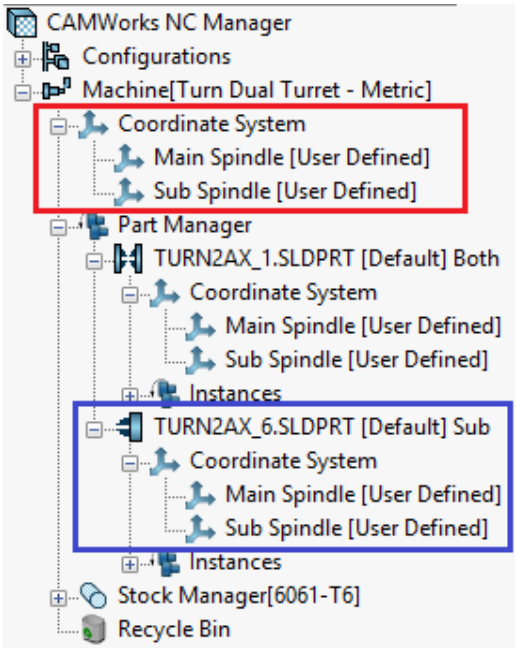


Sample Image of an Assembly comprising Turn and Mill-Turn parts

Defining Coordinate System for Machining the Turn/Mill-Turn Assembly

Use the Machine tab of the Machine dialog box to select the Machine that will be used to machine the part models constituting the Turn/Mill-Turn assembly. Users will have to define the Main Spindle and (optionally) Sub Spindle Coordinate system for this machine. These Coordinate Systems will be used to define the Turn axes of the machine. The Turn axes defined for both the spindles must be co-axial. Use the ***Edit Definition*** button in the ***Setup*** tab of the ***Machine*** dialog box to invoke the ***Main Spindle/ Sub Spindle Coordinate System*** dialog box. Alternatively, you can double-click on the ***Coordinate System*** node under expanded ***Machine*** node in CAMWorks Feature tree and Operation tree.

For each part model, the origin of the Main Spindle and (optionally) Sub Spindle Coordinate System must be separately defined. The origin can be defined using an entity, part vertex or stock vertex. You can double-click on the *Coordinate System* node under expanded Turn part nodes listed under *Part Manager* node in CAMWorks Feature tree and Operation tree to invoke the *Main Spindle/ Sub Spindle Coordinate System* dialog box.

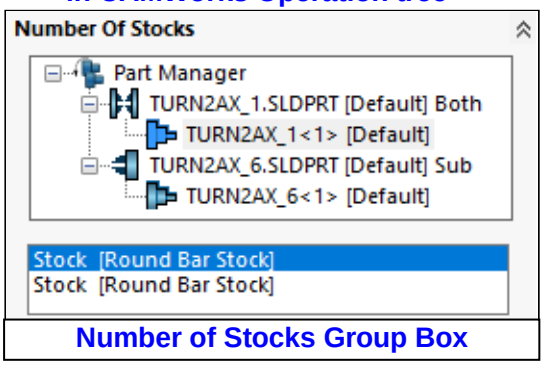


Coordinate System Nodes under Machine node and Turn Part nodes in CAMWorks Operation tree

Defining Stock for the Assembly

Stocks must be defined for each part that constitutes the assembly. The stock for the individual parts will be listed as sub-nodes under Stock Manager node in the CAMWorks Feature tree/ Operation tree. Double-clicking on the any of these ***Stock Manager*** nodes displays the ***Stock Manager*** dialog box.

Use this dialog box to define/edit the stock. Use the ***Number of Stocks*** group box within the ***Stock Manager*** dialog box to select the part model for which stock is to be defined.

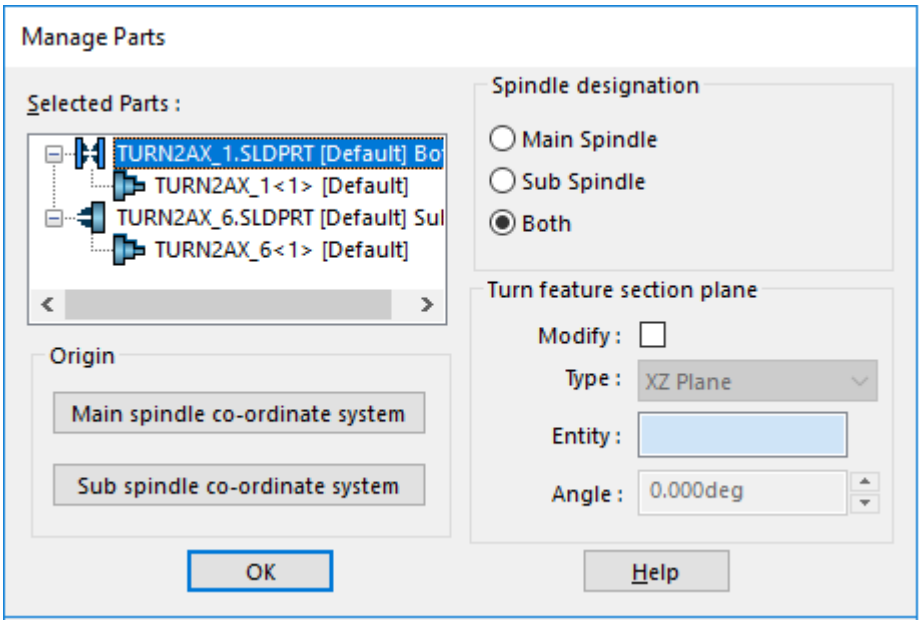


Number of Stocks Group Box

Manage Parts

The Manage Parts dialog box is displayed when you double-click on the ***Part Manager*** node in the Feature tree/Operation tree. This dialog box allows you to:

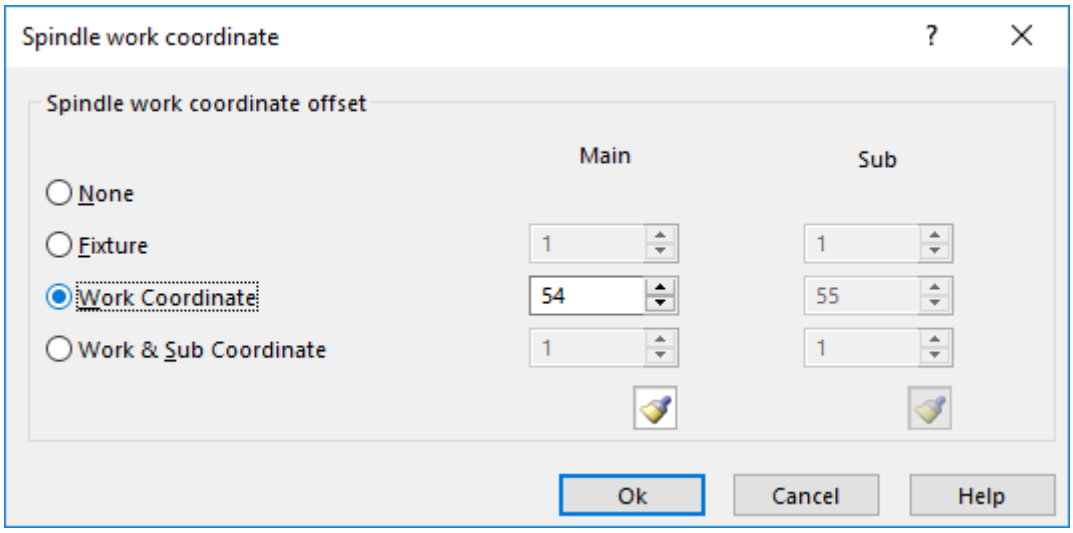
i. Select the Turn/Mill-Turn parts of the active assembly to be machined
ii. Assign the Spindle Designation for each selected part
iii. Assign the Origin of the Coordinate System associated with the Main Spindle/ Sub Spindle for each selected part
iv. Edit the Turn Feature Section Plane



Manage Parts Dialog Box

Defining Programmable Offsets for Main Spindle & Sub Spindle of Active Machine

Programmable coordinate offsets will have to be specified for the Main Spindle and Sub Spindle of each Turn/Mill-Turn part in the assembly.
Click on the ***Edit*** button within the ***Work Offset*** group box under ***Setup*** tab of ***Machine*** dialog box to view the ***Spindle Work Coordinate*** dialog box. Use the parameters within this UI to specify/edit the programmable offsets Main Spindle and optionally Sub Spindle of the active machine.
Work Coordinate will be the default method for defining the offsets. For each part in the Turn/Mill-Turn Assembly, the default Main Spindle value will be set to 54 and the default Sub Spindle value will be set to 55. These default values can be reassigned to suit your requirements.



Spindle Work Coordinate Dialog Box

New - Option to Define the Spindle Direction as Clockwise for Turn and Mill-Turn Machines

Purpose:
To provide a mechanism whereby users can set the Spindle Direction (for Turn and Mill-Turn machines) as Clockwise relative to the Machine Operator

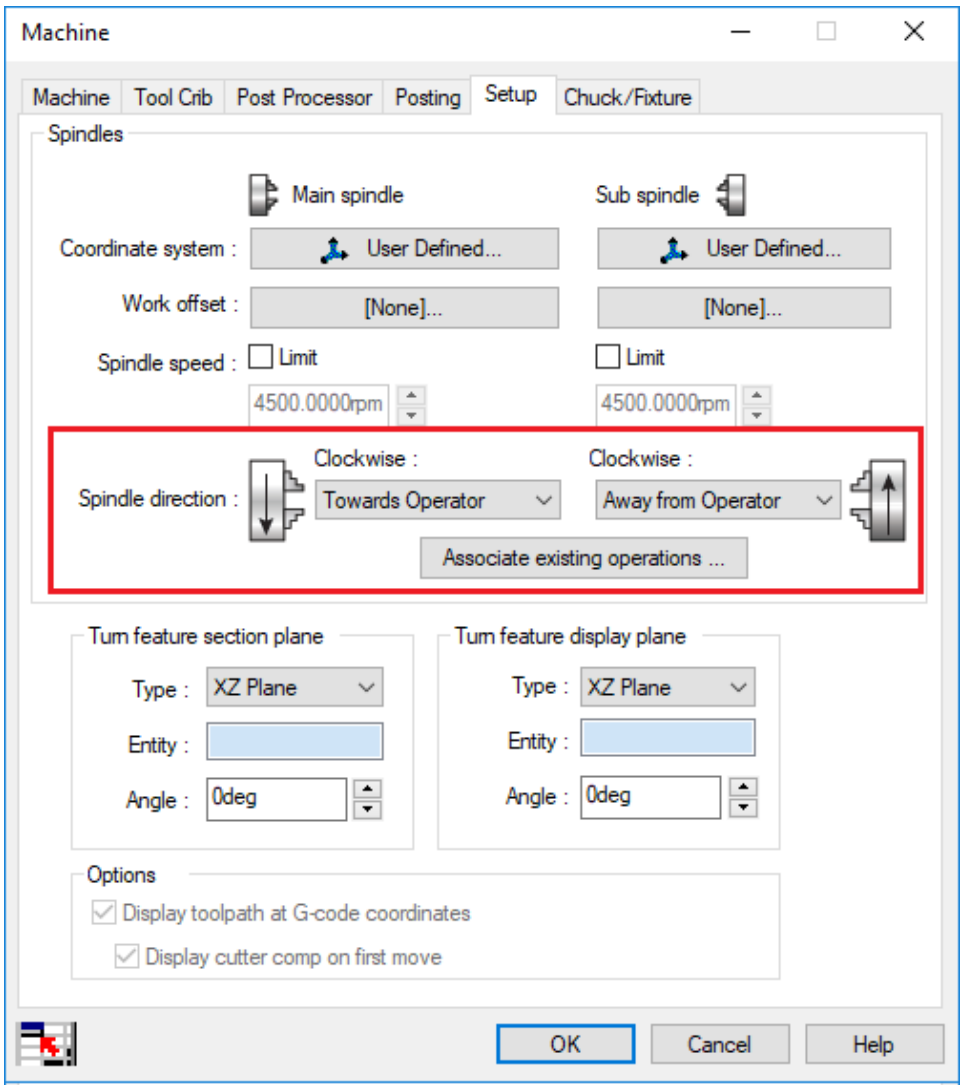
Implementation:

From ***CAMWorks 2021*** version onwards, a new functionality that enables users to set the Spindle Direction as Clockwise for the Main and Sub Spindles of the active Turn/Mill-Turn machine has been introduced. The reference to set the spindle direction as clockwise will be relative to the Machine operator.

Functionality to Define the Spindle Direction as Clockwise for Turn & Mill-Turn Machines

This functionality is provided in the form of the ***Clockwise Spindle Direction*** dropdown list for Main Spindle and Sub Spindle within ***Spindles*** group box of the ***Setup*** tab in ***Machine*** dialog box.

- If the clockwise rotation of the Main spindle and/or Sub spindle is the direction that rotates away from the operator, then select the ***Away from Operator*** option in the corresponding dropdown list.
- If the clockwise rotation of the Main spindle and/or Sub spindle is the direction that rotates towards the operator, then select the ***Towards Operator*** option in the corresponding dropdown list.

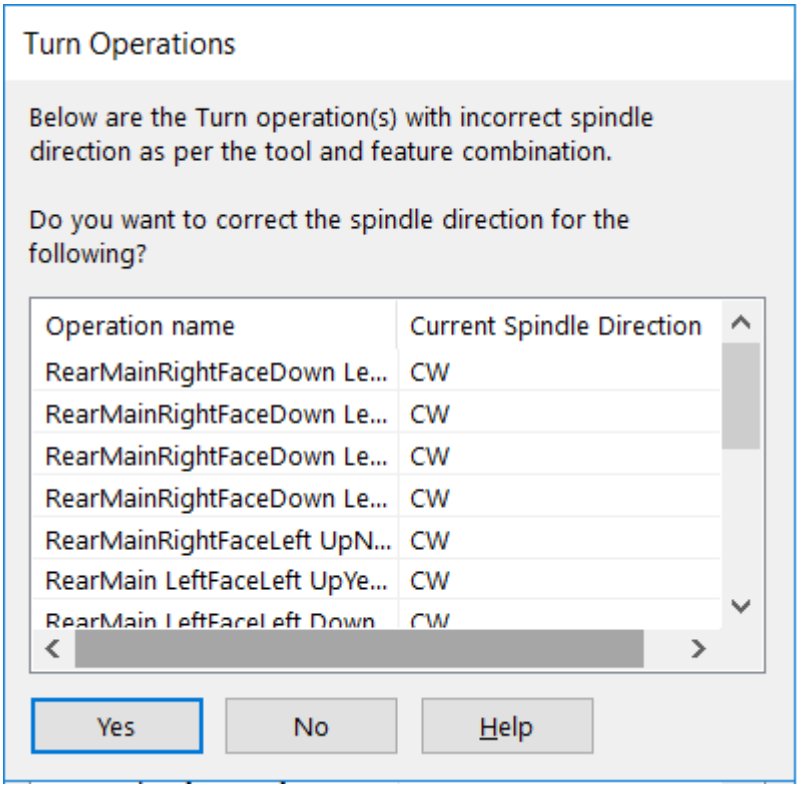


Spindles Group Box within Setup Tab of Machine Dialog Box

Resetting the Spindle Directions of Turn Operations for Legacy Turn/Mill-Turn Parts

If you wish to reset the user-assigned spindle directions of Turn operations for legacy parts, then click on the ***Associate Existing Operations*** button in the ***Setup*** Tab of ***Machine*** dialog box. Clicking on this button displays the ***Dialog box for Turn Operations with Incorrect Spindle Directions***. This dialog box lists all Turn operations for which the spindle directions are with reference to settings made on the machine node.

- Click on the ***Yes*** button within this dialog box to rectify the spindle directions of the listed Turn operations. This action will auto-assign the Spindle direction in the ***F/S tab*** for the Turn operations based on parameters that control the spindle direction. The ***Override Spindle Direction*** checkbox in the ***F/S*** tab of operation parameters will be unchecked.
- If you click ***No*** within this dialog box, then the user-assigned spindle directions for all the Turn operations listed within the dialog box will be retained. The ***Override Spindle Direction*** checkbox option in the ***F/S*** tab of operation parameters will continue to remain in checked state for these Turn operations.



Dialog Box for Turn Operations with Unsynced Spindle Directions listing the name of Turn Operations with Spindle Directions that are not in sync as per the Tool and Feature Combination

New - Mutual Exclusivity of Order of Operations listed in Sync Manager & Operation Tree

Purpose:

i. Option to re-position operations in *CAMWorks Sync Manager* user interface independently without restrictions imposed by operation sequence in Operation tree

ii. Provide command buttons to execute *Step Through toolpath* and *Post Processing* from the *CAMWorks Sync Manager* user interface

Implementation:

For part models and assemblies machined using dual-turret or multi-turret Turn/Mill-Turn machines, the *CAMWorks Sync Manager* functionality allows you to synchronize the toolpaths machined by the tools in the front and rear turrets. This is achieved by re-ordering the operations in the *Sync Manager* tab of the *CAMWorks Sync Manager* user interface. Optimal synchronization can drastically reduce the time-required to machine the parts/assemblies

In previous versions of CAMWorks, the flexibility is re-ordering operations machined using tools in front and rear turrets was limited within the *CAMWorks Sync Manager* user interface as the sequence of operations was linked to the sequence of operations and setups in the Operation tree. For optimal operation sequencing, users had to manually split and/or reorder setups. This often resulted in bloated Setups and a confusing sequence of operations.

From ***CAMWorks 2021 SP0*** version onwards, operations can be re-sequenced in the *Sync Manager* tab of the *CAMWorks Sync Manager* user interface independent of their order in the Operation tree. When the *Syncing Strategy* is set to *User Defined*, all the operations listed from the CAMWorks Operation tree will be listed in their respective turret columns within the *Sync Manager* tab. To reorder the sequence of machining for syncing operations, you can drag and drop the operations to the desired locations in the turret columns as follows:

- By dragging and dropping a selected operation to the desired location within its turret column
- By dragging and dropping it to the specific desired location of another turret column (After re-location, the toolpath of the selected operation will be regenerated.)

Note:

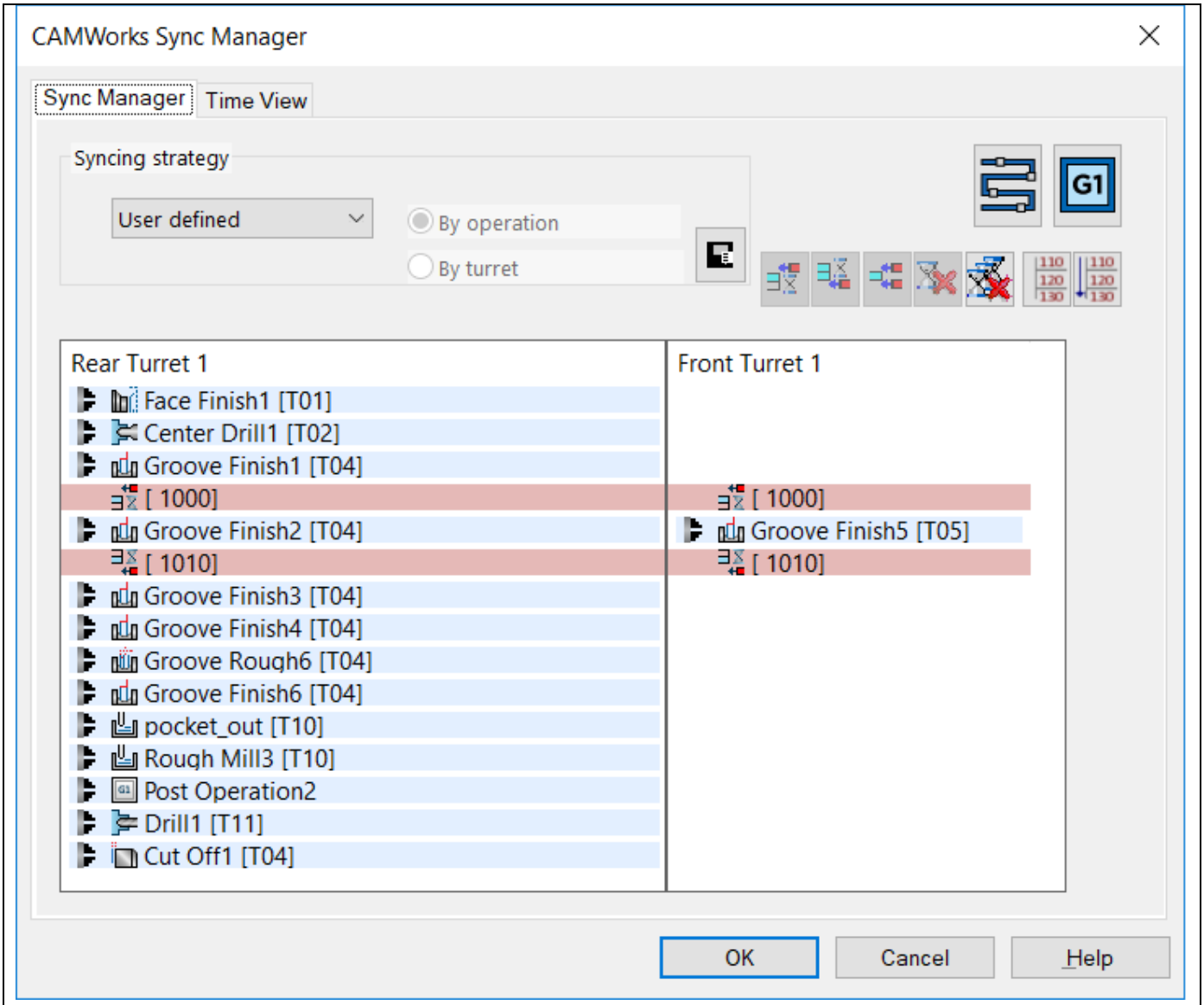
If the position of any operation in the turret column is changed, then all the toolpaths for the operations listed in the turret columns will be regenerated without any warning message being displayed. This behavior is observed because, by default, the ***Update toolpaths when parameters change*** option in the ***Update*** tab of ***CAMWorks Options*** dialog box is set to '***Always***'. To receive a warning message regarding the recalculations of toolpaths when operation parameters change, set the ***Update toolpaths when parameters change*** option to '***Prompt***'. This setting ensures that whenever the position of an operation in the turret is changed, CAMWorks will display a warning message regarding the need to regenerate toolpaths. Click ***Yes*** within the warning message to regenerate the toolpaths.

Executing Step Through Toolpath Command  from CAMWorks Sync Manager User Interface

The ***Step Through Toolpath*** command is now available in the *CAMWorks Sync Manager* user interface. If you execute this command, then the *Step Through Toolpath* dialog box will be displayed. Use this user interface to step through the toolpaths. The toolpath sequence for stepping through the toolpaths will be based on the order of operations listed in the *Sync Manager* tab. It will not be based on the operation sequence listed in the Operation tree.

Executing the Post Process Command  from CAMWorks Sync Manager User Interface

The ***Post Process*** command is now available in the *CAMWorks Sync Manager* user interface. If you execute this command, then the NC program generated will be based on the order of operations listed in the *Sync Manager* tab. It will not be based on the operation sequence listed in the Operation tree under various Turn or Mill Setups.



Sync Manager Tab of CAMWorks Sync Manager User Interface

Feed & Speed Library

Improved - Updated 64-bit Feed and Speed Library now Shipped with CAMWorks

Purpose:

To provide an enhanced version of the Feed and Speed Library with SQL-based database engine to users

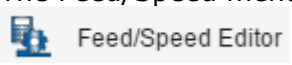
Implementation:

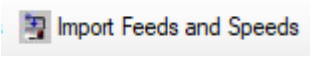
In previous versions of CAMWorks, a 32-bit version of the *MES Feed and Speed Library* was shipped along with CAMWorks. This application had a MS Access based Database Engine.

From ***CAMWorks 2021 SP0*** version onwards, the 64-bit version of the ***MES Feed and Speed Library*** will be shipped in the CAMWorks Installer Package. This application has an SQL-based database engine. For users who wish to further customize the entries present within the *Feed and Speed* library, the user interface too has undergone a makeover that makes the navigation and interaction intuitive and smooth.

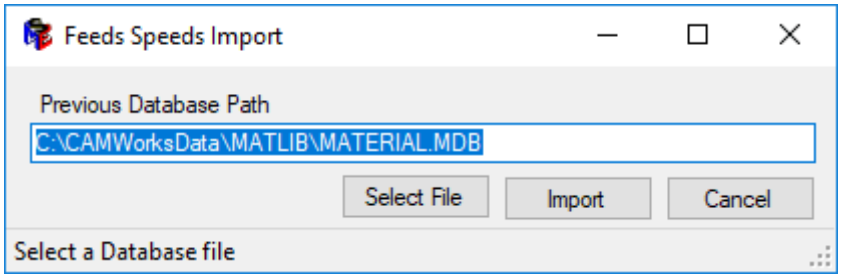
If you had customized feed and speed data in the 32-bit Feed and Speed Library, then you will need to import it into the 64-bit *Feed & Speed Library*. Given below are the steps for importing customized data.

Steps to Import Customized Feed & Speed Data in 64-bit CAMWorks Feed & Speed Library

1. Launch the CAMWorks Technology Database.
2. Click on the *Feed/ Speed* menu  on the left-hand side.
3. The Feed/Speed menu options will be displayed. Click on the first option labelled 

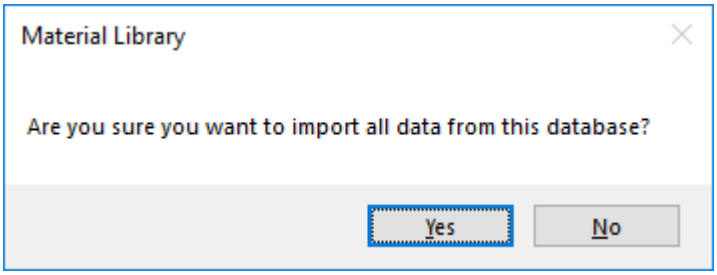
.
4. This action will launch the *CAMWorks Feed and Speed Library*. Click on the 

command on its ribbon bar.
5. A warning message will inform you that all existing data within the *Feed and Speed Library* (including customized data, if any) will be replaced on going ahead with the import process. Click ***Yes*** within the warning message to confirm.
6. As your previous *Feed and Speed Library* version was a 32-bit application with a MS Access based database engine, the current 64-bit *Feed and Speed Library* application (with a SQL database engine) will fail to find the database. A warning message will be displayed stating that no data was found and that you manually need to enter the path to the database of the previous version. Click ***OK*** within this message box.
7. The *Windows File Explorer* will be launched. Use it to navigate to the *.mdb file (MS Access database) of the previous *Feed and Speed Library* application. The default path to the 32-bit *CAMWorks Feed and Speed Library* application is:
C:\CAMWorksData\MATLIB\MATERIAL.MDB
8. Select the *Material.MDB* file and execute the *Open* command.
9. The ***File Explorer*** will close and the *Feeds Speeds Import* window will be displayed. This window indicates the path to the database file from which Feed & Speed data will be imported. Click on the *Import* button within this window to commence importing.



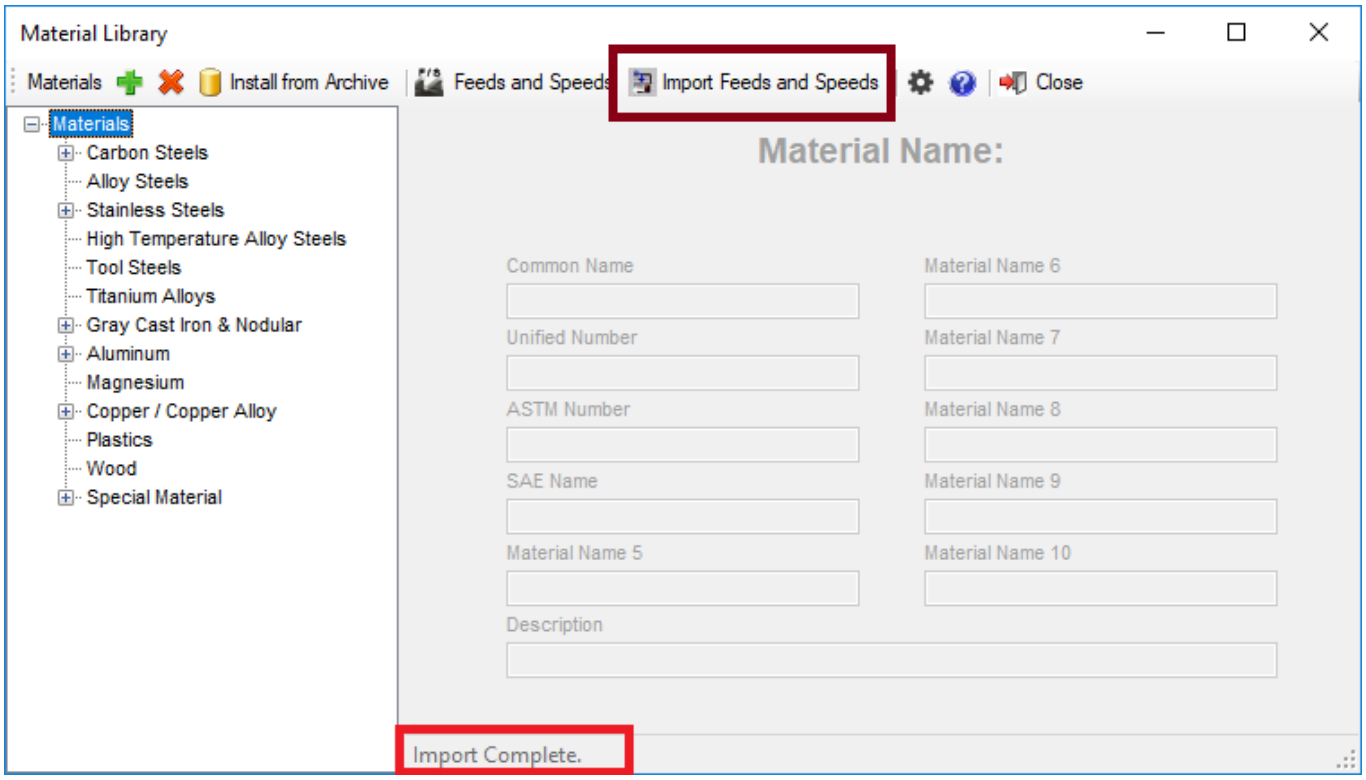
Feeds Speeds Import Window

10. A warning message will prompt you confirm whether you are sure about importing all data from the selected database. Click ***Yes*** within this message box.



Warning Message Prompt to Confirm Import Process

11. The import process will commence. The user interface will revert to the ***Feed and Speed Library*** application. Observe that in the ribbon bar, the Import Feeds and Speeds button command has been temporarily replaced with the Importing Now... command (in disabled state). Depending on the volume of the data present, this process will take several minutes.
12. Once the import is complete, a message indicating successful import of the data will be displayed in the status bar of the user interface. The Importing Now... button will be replaced with the Import Feeds and Speeds button.



Status of Import Process indicated in Status bar

13. Your 64-bit SQL based database engine *Feed and Speed Library* now contains all your customized data. The next time you use TechDB or retrieve information from the TechDB while using the CAMWorks application, the Feed/ Speed values applied will be based on the customized data you imported in to the 64-bit SQL based database engine *Feed and Speed Library* application.

ShopFloor Publisher

New - Option to copy/reference additional documents in a ShopFloor File to be Published

Purpose:

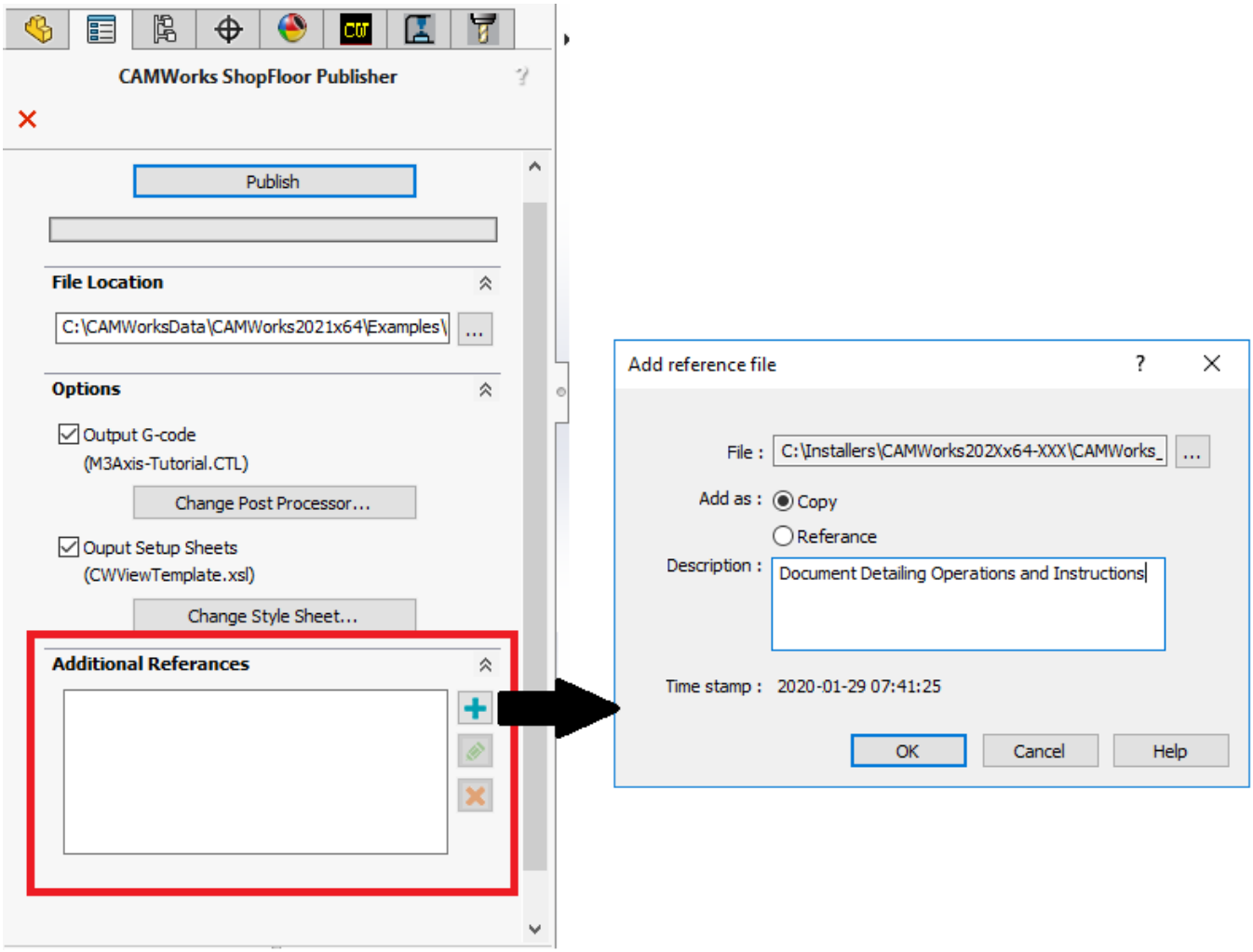
To provide the option to copy/reference additional documents in a ShopFloor file to be published

Implementation:

From ***CAMWorks 2021 SP0*** version onwards, you can provide additional files in any file format as a copy or file reference within the published ShopFloor file.

Use the parameters available within the newly introduced ***Additional References*** group box available in the ***CAMWorks ShopFloor Publisher*** dialog box for this purpose. Use the parameters within this group box to:

- Add a copy of one or more files of choice
- Create references to one or more files of choice
- Change any file copies or references



'Additional References' group box in CAMWorks ShopFloor Publisher Dialog Box

Clicking on the ***Add New File*** button displays the ***Add Reference File*** dialog box. Use this dialog box to add a copy of a file of choice or provide a reference path to it in the ShopFloor file you intend to publish. Once the ***Add Reference File*** dialog box is closed, the file path to the selected file will be displayed in the ***Additional References*** list box.

The ***Delete Selected File***  command button will be enabled only when a file path listed in the ***Additional References*** list box is selected. Click on this command button if you wish to delete the selected file path reference.

ShopFloor Document Manager

New - ShopFloor Document Manager Application for Machine ShopFloor Personnel

Purpose:

To provide a zero extra cost application for Machine ShopFloor personnel that enables them all necessary information for machining parts and assemblies within a single user interface

Implementation:

ShopFloor Document Manager Application

The ***ShopFloor Document Manager*** is a newly introduced **zero-cost** standalone application now available in the CAMWorks Installer Package. It can be installed and run on any Windows system, even those on which other *CAMWorks* products are not installed.
It serves as a collaboration tool between the following sets of personnel:

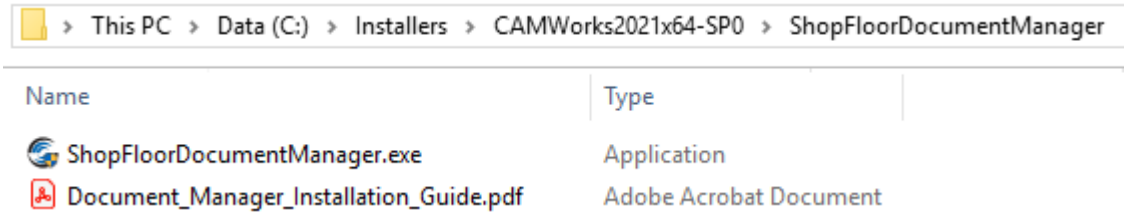
- **CAM Programmers** (who use the ***CAMWorks*** application to generate CAM data for part models and/or assemblies to be machined)
- **Machine ShopFloor Personnel** (who execute the actual machining of the concerned part models/assemblies at the Machine ShopFloor)

Intended users of ShopFloor Document Manager Application

The Machinists working on the Machine ShopFloor are the intended users of the ***ShopFloor Document Manager*** application.

Installing the ShopFloor Document Manager Application

The installer for the ***ShopFloor Document Manager*** application is available within the ***CAMWorks 2021x64*** installer package within the ***ShopFloorDocumentManager*** sub-folder.



ShopFloor Document Manager Installer within CAMWorks Installer Package

Current Collaboration Process followed by CAM Programmers & Machinists

i. Once the CAM Programmer fully programs the part model/assembly using ***CAMWorks*** (i.e. the toolpaths have been verified and G-code has been generated), he/she generates the Setup Sheets and G-code file using associated commands in the application.
ii. The Setups Sheets and G-code file are then shared with the Machine ShopFloor personnel.
iii. The Setup Sheets are used for referring information on how to set up the stock, the machine, the controller, estimated machine time, part material, and the tooling used to machine the part. Optionally, the Setup Sheets may also include WIP images, Part/assembly images and/or Tool images.
iv. The Machine ShopFloor personnel use 3rd party-software to view/edit the G-code file.
v. After verifying the G-code, the part model/assembly is machined as per specifications.

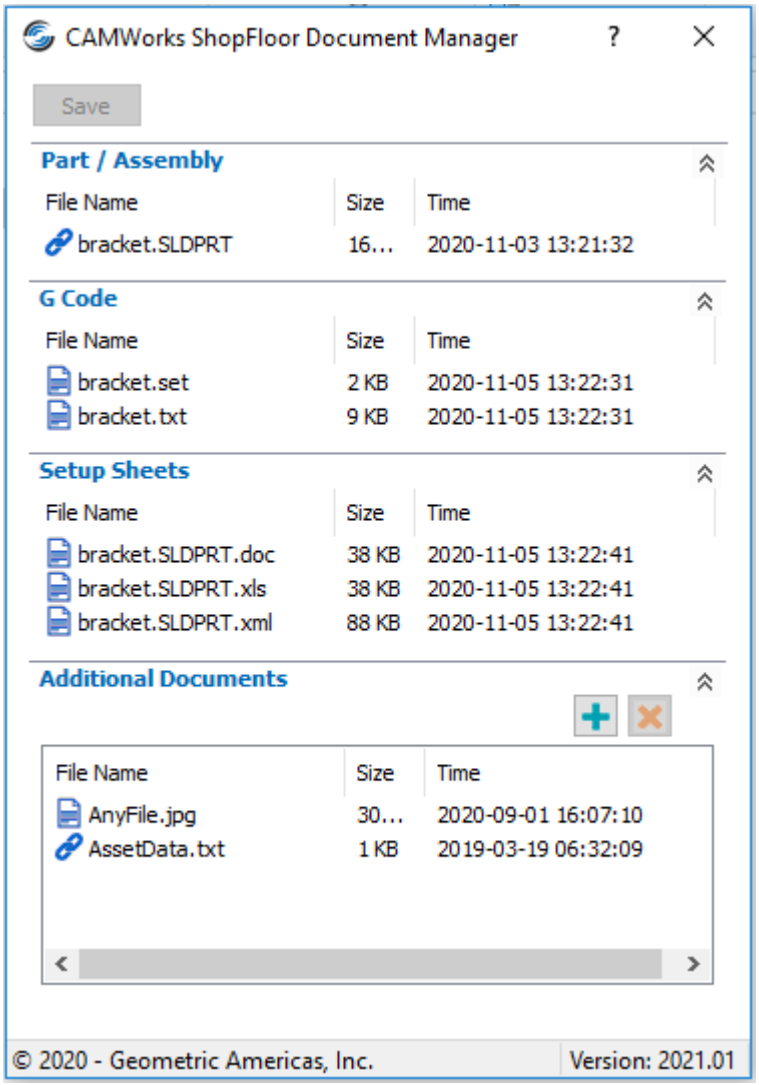
How the Collaboration Process will work with CAMWorks + ShopFloor Document Manager Combination

The *CAMWorks* application contains a *Publish ShopFloor* functionality that allows CAM programmers to publish a *ShopFloor* file for the part model/ assembly to be machined.
This ***ShopFloor*** file (with **. cwspflr* file extension) serves as a digital container for the following manufacturing data required by Machine ShopFloor personnel for machining part models/ assemblies:

- Data files required for CAD visualization (Part Model/Assembly view)
- CAM data
- Toolpath Simulation data
- G-code
- Setup Sheets

Such a published ***ShopFloor*** file can then be shared with the Machine ShopFloor personnel. The Machine ShopFloor personnel can then use the ***ShopFloor Document Manager*** application to view info contained in the ShopFloor file.

On launching the ***ShopFloor Document Manager*** application. A ***File Explorer*** window will prompt the user to select the ShopFloor file to be viewed. Once the file is selected, the *ShopFloor Document Manager* user interface will display information contained within the ShopFloor file.



User Interface of the ShopFloor Document Manager

Value Proposition offered by Publishing & Viewing ShopFloor Files

- All the manufacturing data required by the Machine ShopFloor personnel is made available in a digital format (****. cwspflr***) within a single user interface (viz. the UI of the ***CAMWorks ShopFloor*** application.)
- The association of the manufacturing data with the part model/assembly is maintained
- No need for an expensive ***SOLIDWORKS*** license for viewing the part model & associated CAM data [Either the ***ShopFloor Document Manager*** (a freeware application) or the ***CAMWorks ShopFloor*** (application with cost-effective license) can be used].