CAMWorks®
A Geometric Product

What's New in CAMWorks 2017

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

October 2016

What's New in CAMWorks 2017 – SP0

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2016 - CAMWorks Solids 2016 - SOLIDWORKS 2017 SP0 **Note:** CAMWorks Solids 2017 will be supported in the future releases.
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] **Note:** CAMWorks 2017 is supported only on 64-bit Operating systems.

Resolved CPR's 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 (PR1).
Implementation:	To view this document, click on the Windows *Start* menu and select: *All Programs→CAMWorks2017x64→Resolved CPR's*

Minimum Supported Version of MS Access for sharing TechDB on a network drive

Supported Versions of MS Access:	Microsoft Access is a pre-requisite for using the Technology Database. The following 32-bit and 64-bit versions of Microsoft Access are supported: • MS Access 2010 SP2 (32-bit and 64-bit versions) • MS Access 2013 SP1 (32-bit and 64-bit versions) • MS Office 365 (32-bit and 64-bit versions) • MS Access 2016 (32-bit and 64-bit versions)

Minimum Supported Version when sharing TechDB on a network:

The minimum compatible version of MS Access required when sharing TechDB/ Report Database over a network drive is ***MS Access 2010 SP2*** (Runtime or Full). If you do not have this minimum supported version installed, you can install the *MS Access 2010 SP2* Runtime version from the ***AccessRuntime*** sub-folder of the *CAMWorks Installer Package*.

Note:

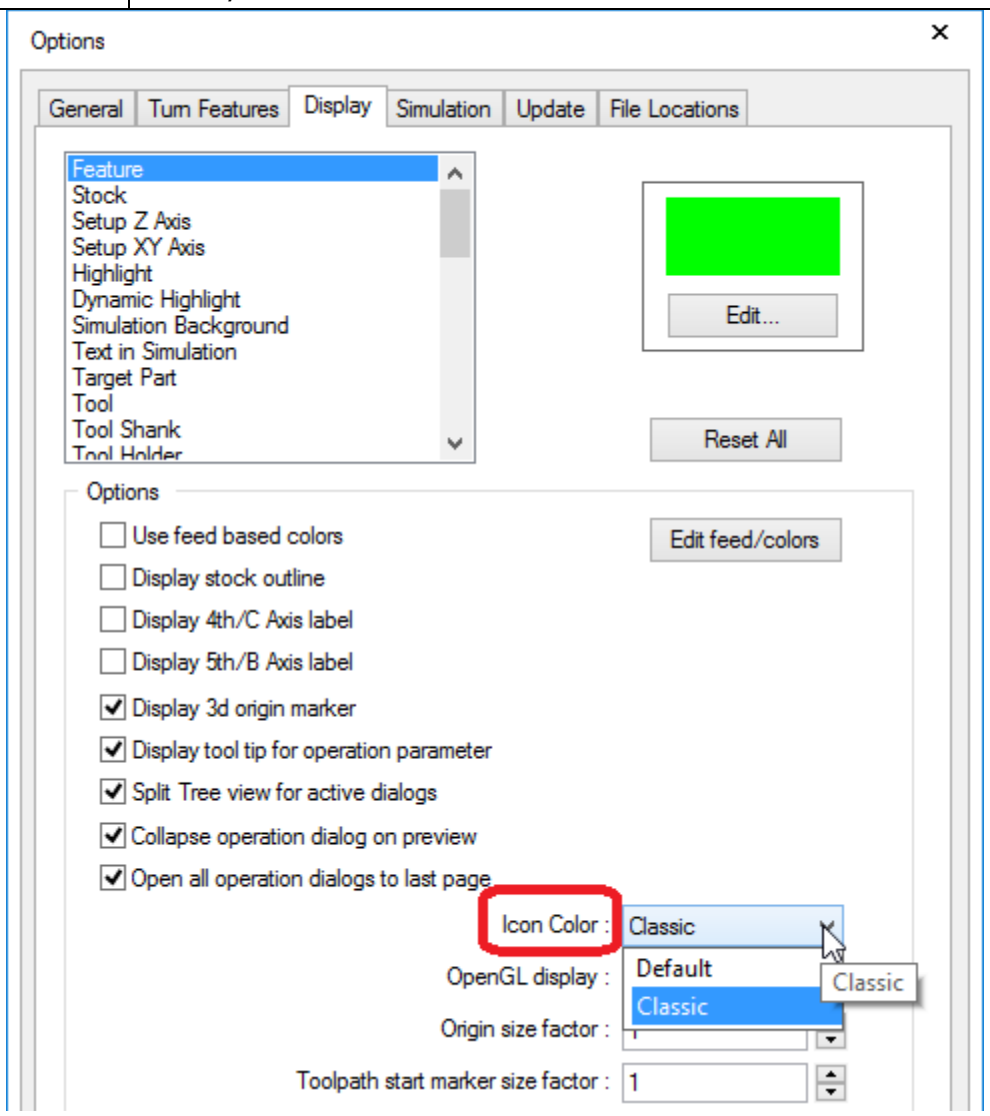
MS Access 2007 version is still supported but is not recommended when TechDB is shared over a network.

When CAMWorks is launched as an Add-In within *SOLIDWORKS/ CAMWorks Solids* with TechDB shared on a network, CAMWorks detects the MS Access version. If the MS Access version is lower than *MS Access 2010 SP2*, then a warning message prompting the user to install *MS Access 2010* or higher version will be displayed.

General

New - Default and Classic color schemes for icons in the CAMWorks User Interface

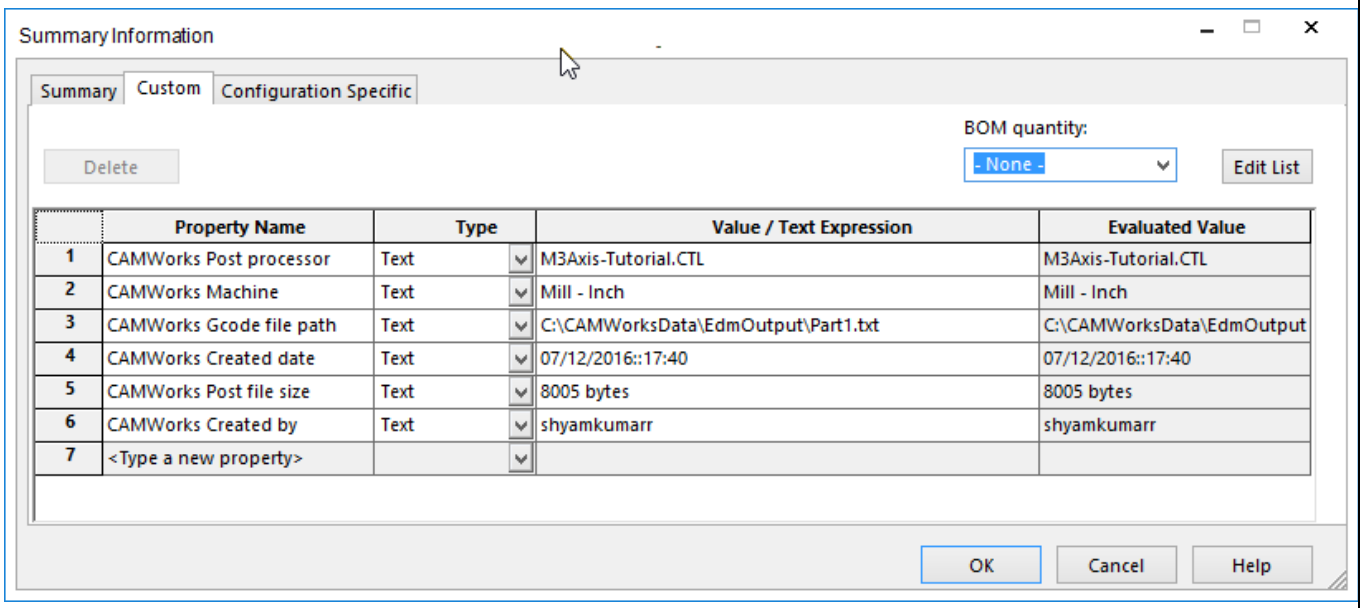
Purpose:	Offers user the choice for color scheme of CAMWorks icons
Implementation:	From ***CAMWorks 2017*** version onwards, two color schemes viz. ***Classic*** and ***Default*** are offered in CAMWorks. This color scheme applies to all the icons associated with CAMWorks such as the icons in the Command Manager, Feature tree, Operation tree, context menus, pull-down menus and dialog boxes. This setting is available in the ***Icon Color*** dropdown list of the ***Display*** tab of ***CAMWorks Options*** dialog box. The color scheme ***Default*** is the default color scheme at the time of installation. After toggling the schemes, the changes will be effective only when you restart CAMWorks.



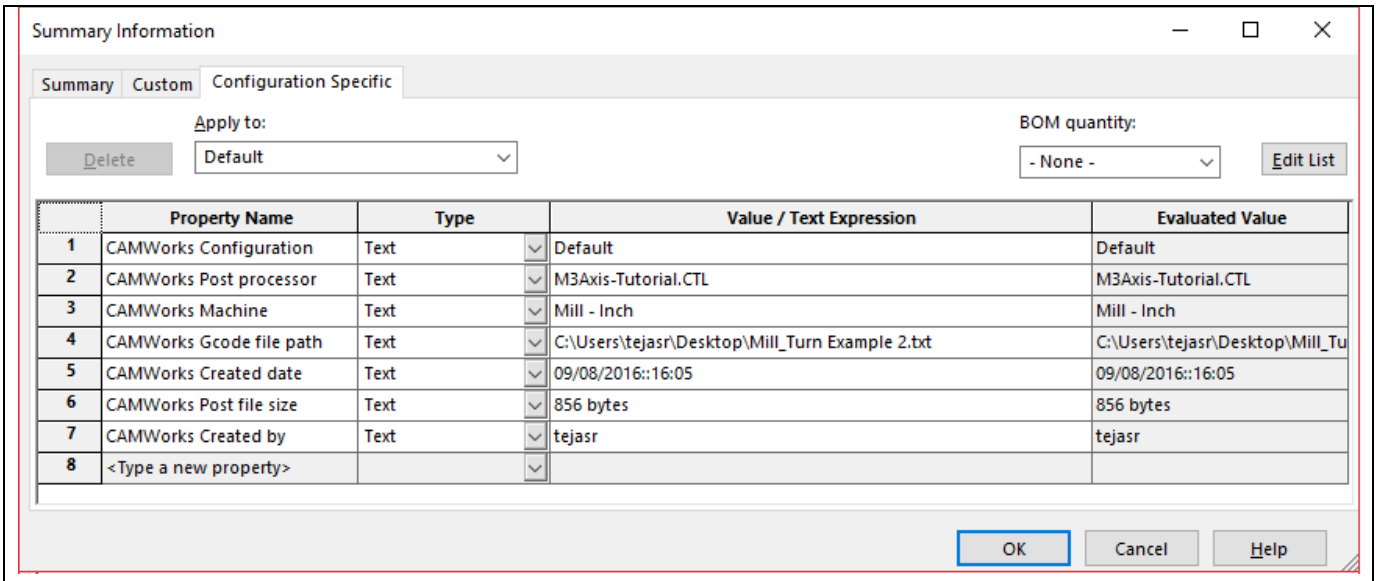
'Icon Color' parameter in Display tab of CAMWorks Options dialog box

New - Viewing Post Processing Data Info in SOLIDWORKS Properties

Purpose:	Provides the ability to view CAMWorks information generated for a part/assembly as Custom properties of a SOLIDWORKS part/assembly file
Implementation:	The properties of a SOLIDWORKS part/assembly can be viewed by clicking on the SOLIDWORKS ***File*** menu and selecting ***Properties***. Information related to CAM data generated using CAMWorks is added to the ***Custom*** tab of the ***Summary Information*** dialog box after the post processing the toolpaths. If the SOLIDWORKS part or assembly file has more than one configuration or if CAMWorks also has corresponding configurations, then CAM data generated using CAMWorks will be stored with the associated SOLIDWORKS configuration. Such configuration specific CAM post-related properties will be displayed in the ***Configuration Specific*** tab of the ***Summary Information*** dialog box when the part/assembly file is post processed. Following CAMWorks Post-related properties are displayed in these tabs: **Property Name** — **Parameter name** • CAMWorks Post Processor - Name of the Post Processor • CAMWorks Machine - Name of the Machine • CAMWorks G-Code File path - Last posted file name along with File location path • CAMWorks Created Date - Date when G-Code while was post processed • CAMWorks Post File Size - Size of the post processed file • CAMWorks Created by - The name of the programmer • CAMWorks Configuration - Name of the CAMWorks Configuration **Note:** These parameters will be overwritten every time the toolpaths are post processed. After *Post Process* command is executed, manual editing of CAMWorks Properties listed in this tab is possible. Changes thus made can be saved as well.



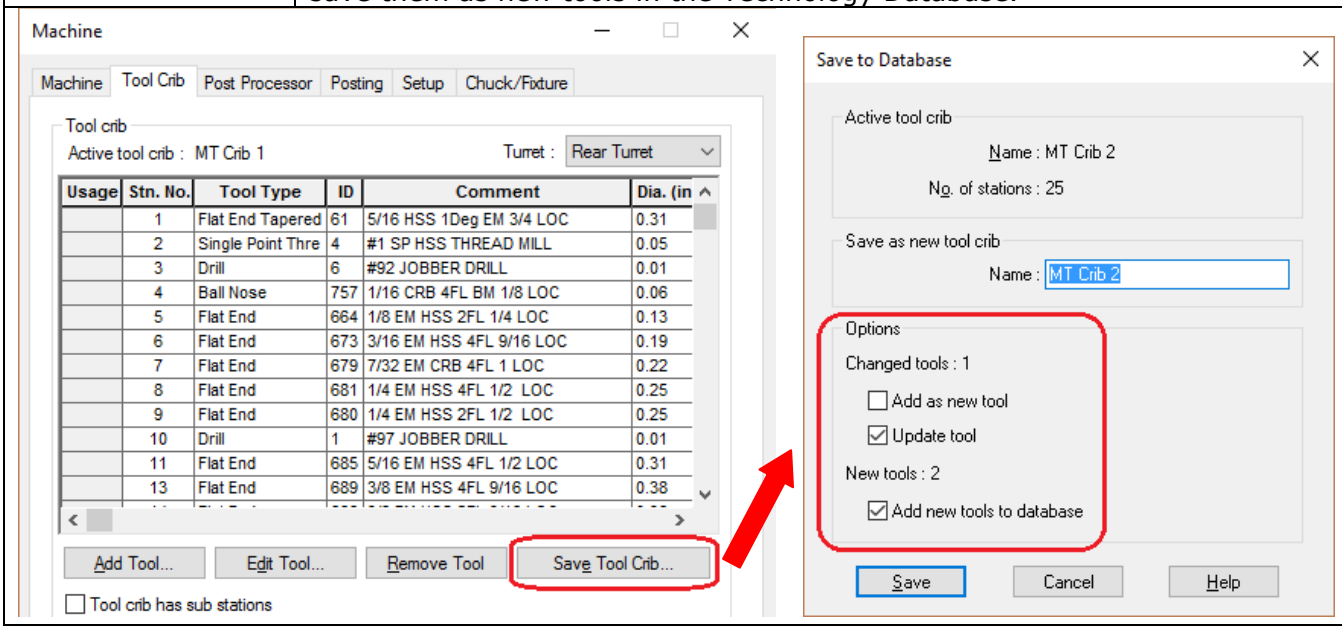
Custom tab of Summary Information dialog box displaying CAMWorks Properties



Configuration Specific tab of Summary Information dialog box displaying CAMWorks Properties

Improved - Save tools while saving changes made to Active Tool Crib

Purpose:	Provide new options in the ***Save to database*** dialog box (displayed when S***ave Tool Crib*** button in *Tool Crib* tab of *Machine* node is clicked) using which you can either save changes made to existing tools in the Active Tool Crib or save them as new tools in the Technology Database.



Implementation:	**Add as new tool** When the ***Add as new tool*** checkbox option is selected, existing tools in the active tool crib that have undergone modifications will be saved as new tools in the Technology Database on clicking the ***Save*** button. Such new tool will also be added in the active Tool Crib being saved with a new TechDB ID and Station number. **Update tool** When the ***Update tool*** checkbox option is selected, all tools in the Active Tool Crib that have undergone modifications will be saved in the Technology Database with their original TechDB ID and Station number.

	New tools This number indicates the number of new mill tools (which are not part of the TechDB) that have been created in CAMWorks within the current solid part or assembly. **Add new tools to database (option available only for Mill & Mill-Turn machines)** When the ***Add new tools to database*** checkbox option is selected, any new mill tool which has been created in CAMWorks within the current part or assembly will be added to the active tool crib as well as the Technology Database. **Note:** When the tools used to machine the current part or assembly are not available in the TechDB, use this option to add such tools to the TechDB and active tool crib so that they can be referred for future use.

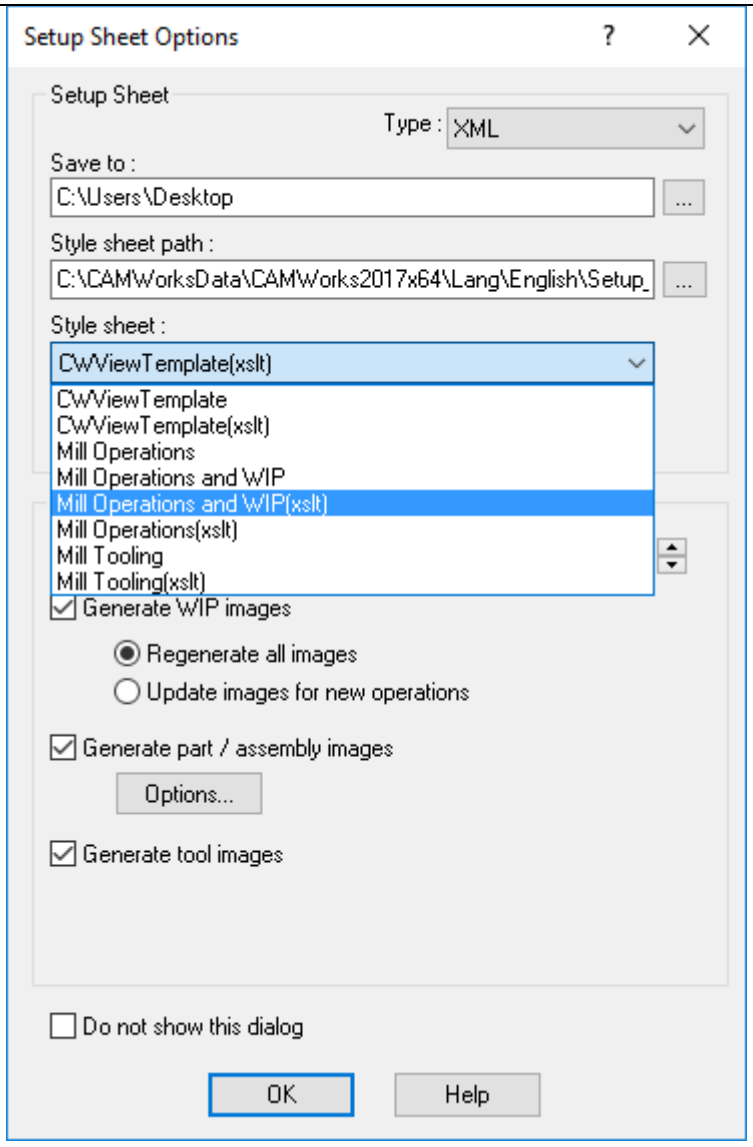
New - XSLT Stylesheet template for viewing XML-based Setup Sheets in MS-Excel

Purpose:	Provides the ability to generate XML-based Setup Sheets which can be viewed in Internet Explorer web browser and are also compatible with MS Excel and can therefore be viewed/edited using MS Excel (Version 2007 or higher).

Implementation:

In Setup Sheet Options dialog box, the XSLT based stylesheets will be available in the Stylesheet dropdown list when the Setup sheet Type is set to XML.
XSLT style sheets in the dropdown list can be easily recognized by the *(xslt)* suffix at the end of the respective style sheet names.

Similar to XML-based Setup Sheets generated using XSL stylesheets, Setup sheets generated using XSLT style sheets too can be viewed in the *Internet Explorer* web browser. However, they are also compatible with Microsoft Excel (Versions 2007 and higher). Therefore, such Setup sheets can be viewed/ edited using MS Excel.



Setup Sheet Options dialog box

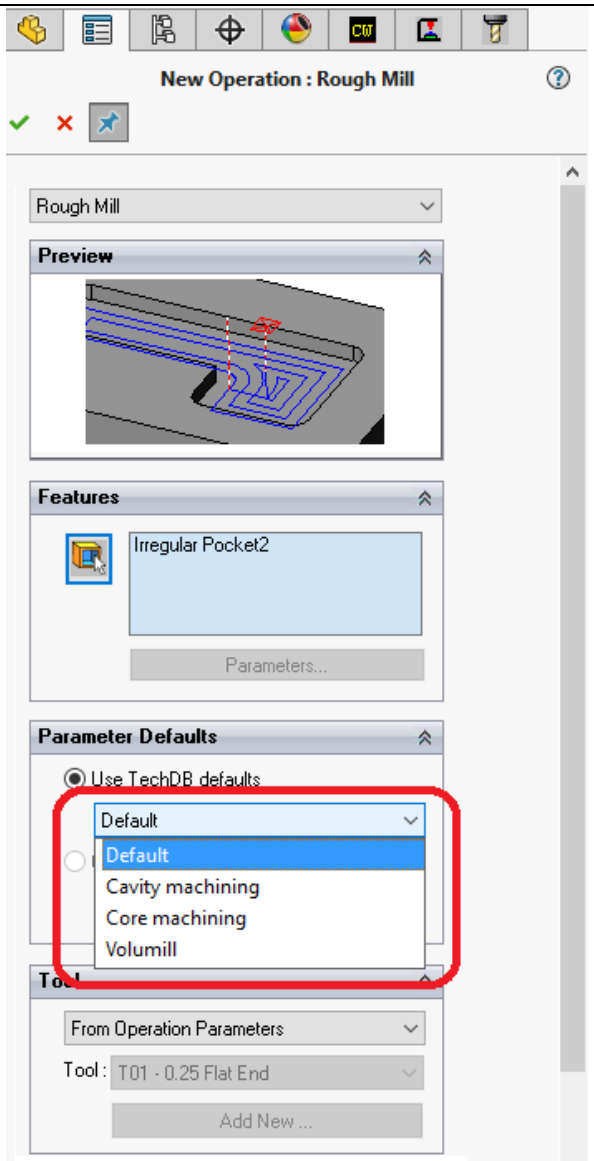
Improved - Ability to choose and define multiple Default Operation Parameters (TechDB)

Purpose:	The User Interface of the ***Default Operation Parameters*** form in TechDB has been simplified to allow the user to select operation type and define the default parameter instances.

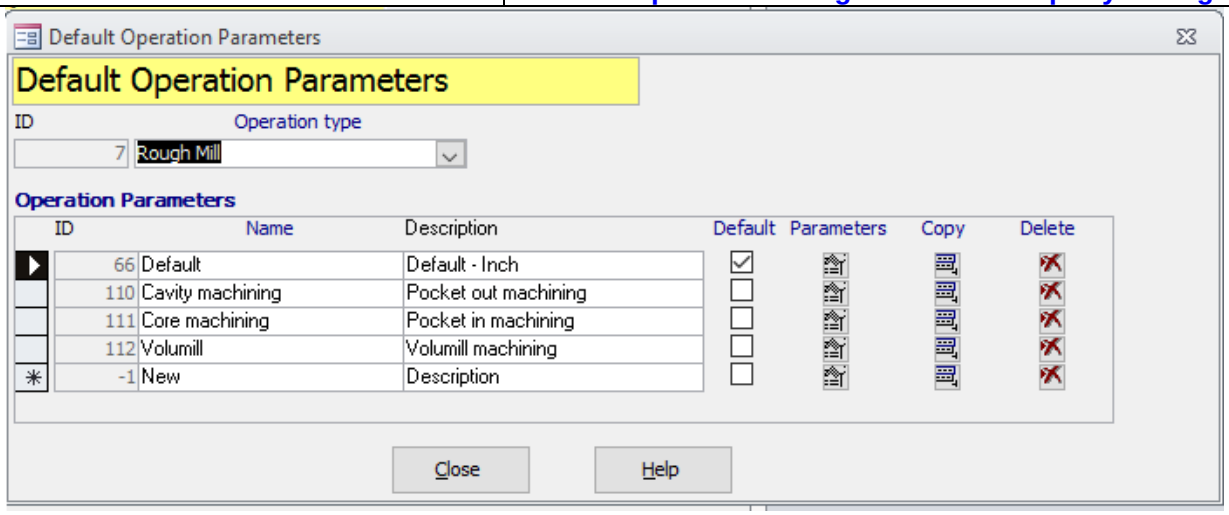
Implementation:

The ***Default Operation Parameters*** form (image given below) is displayed, when the Default Operation Parameters item is selected. The form lists the Operations and default parameter instances that have been defined for each operation type. This form also allows user to define the following changes for each operation type:

- Define new parameter instances
- Edit the parameters of existing instances
- Copy and paste existing parameter instances
- Delete parameter instance
- Select the desired default parameter instance



New Operation dialog window in Property Manager

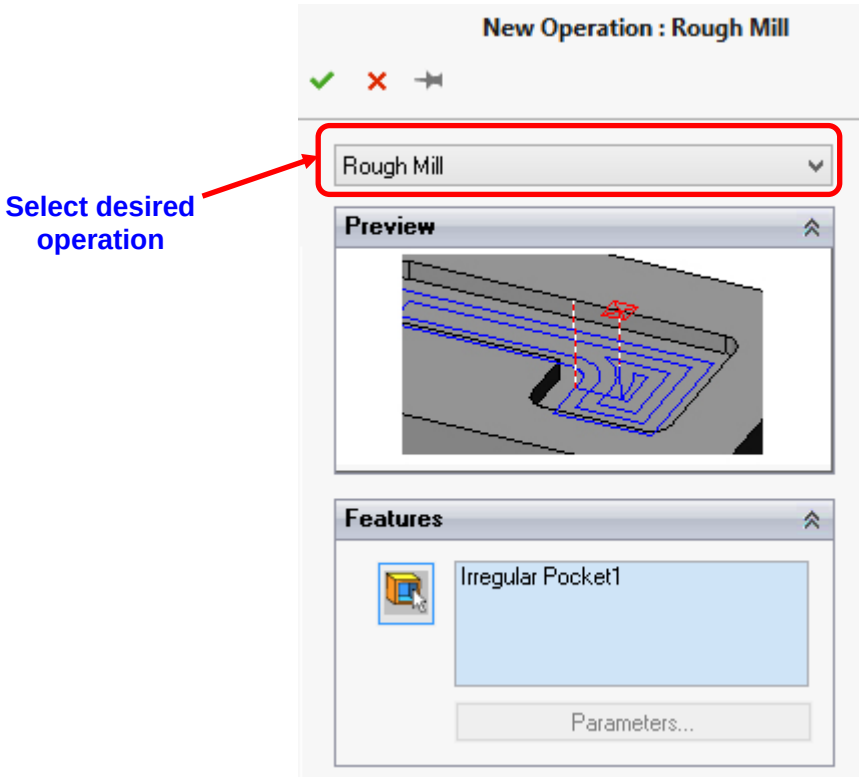


Default Operation Parameters form in TechDB

Modified - Modified Command Manager buttons for Operations

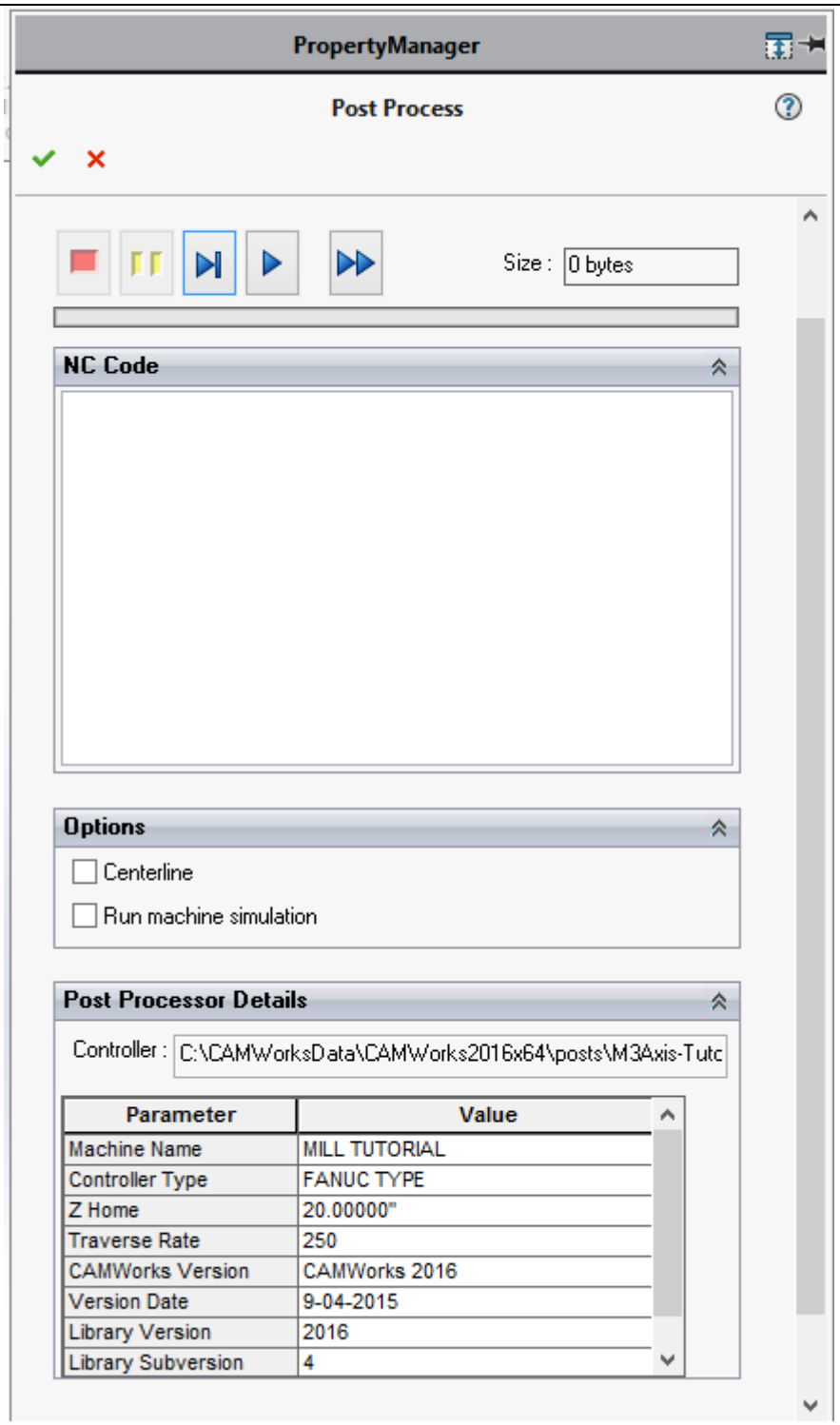
Purpose:	To have a single command in the Command Manager for every type of operation.
Implementation:	From ***CAMWorks 2017*** version onwards, the ***Insert Operation*** commands on the CAMWorks Command Manager have been modified such that the User Interface for *New Operation* is shown in one mouse click. You will be able to choose the desired type of operation within the ***New Operation*** dialog window in the Property Manager tab.

Insert Operations Command on Command Manager tab

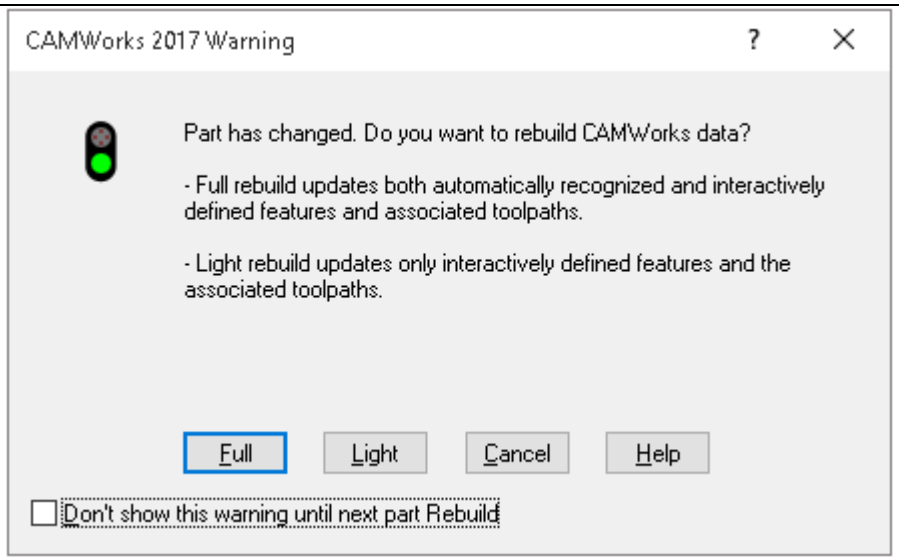


New Operation dialog window in Property Manager

Improved - Improved User Interface for Post Processing in Property Manager Page

Purpose:	Move the parameters in the ***Post Processor*** dialog box to the ***SOLIDWORKS Property Manager*** page to make the User Interface more user-friendly.
Implementation:	 **Post Processor Property Manager Page**

Improved - Improved description within Rebuild Message dialog box

Purpose:	Descriptive text in the Rebuild Warning message dialog box so as to facilitate easier understanding between Full Rebuild and Light Rebuild options.
Implementation:	 **Rebuild Warning Message Dialog Box**

Improved - Improved index numbering sequence for newly created CAMWorks entities

Purpose:	To reset numbering sequence back to '1' when all instances of a specific CAMWorks entity (Setups, Features, Operations, Avoid Areas or Contain Areas) listed in the CAMWorks Feature tree/ Operation tree are deleted and new instances are created
Implementation:	In CAMWorks, whenever new entities such as Features, Operations, Setups, Contain Areas or Avoid Areas are created, they are assigned numbers in order to distinguish between multiple instances of the entity. The numbering is sequential and is assigned as a suffix to the entity's name in the CAMWorks Feature tree/ Operation tree. In previous versions of CAMWorks, when all instances of a specific CAMWorks entity listed in the CAMWorks Feature tree/ Operation tree were deleted and user proceeded to create new instances, the numbering sequence would continue based on the previously deleted list. This ended up creating confusion for users. From *CAMWorks 2017*, if all the instances of any CAMWorks entity listed in the CAMWorks Feature tree/ Operation tree are deleted and new instances are created, then the numbering sequence for new entities will be reset to 1. Numbers will be assigned sequentially to instances in the order of generation. **Note:** Numbering sequence for an entity will not be reset to '1' if one instance of the entity is retained in the Feature tree/ Operation tree.

Mill

New - VoluMill Technology Expert for feed and speed parameters

Purpose:

Provide an easy method to select appropriate cutting parameters for Volumill toolpaths.

Implementation:

Accessing the VoluMill Technology Expert Functionality

Clicking on the ***VoluMill Technology Expert*** button in the F/S tab of a Mill Operation opens the ***VoluMill Technology Expert*** dialog box. This button is enabled only for 2.5 Rough Mill operations and Area Clearance operations when the following conditions are fulfilled:

- VoluMill is selected as the Roughing pattern
- Feeds and speeds are defined by Operation

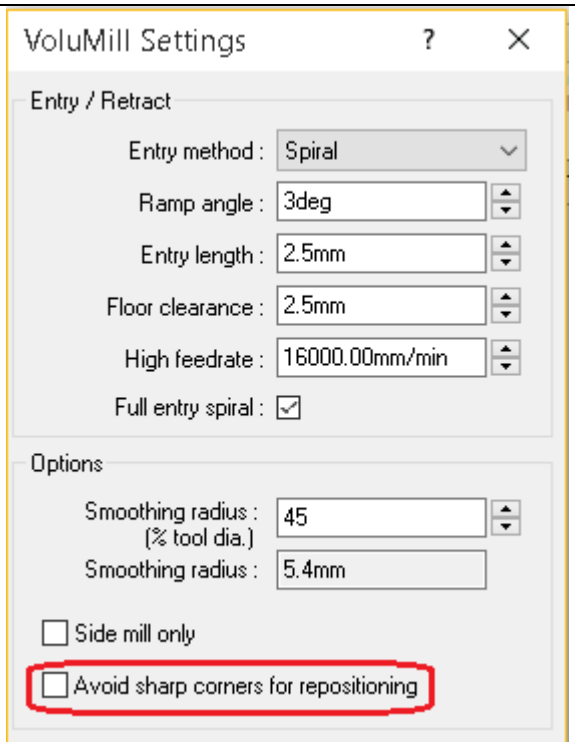
VoluMill Technology Expert dialog Box

The ***VoluMill Technology Expert*** makes use of user-defined parameters to calculate the cutting parameters on tool and stock material combinations. You can choose cutting parameter values that range between conservative and aggressive values. It is especially recommended that you use this functionality for VoluMill operations since the cutting parameters for VoluMill-based toolpaths are considerably different from normal roughing toolpaths.

When you click the ***OK*** button in the ***VoluMill Technology Expert*** dialog box, the cutting parameters displayed in the ***Selected*** column of the ***Recommendations*** group box will be transferred to the respective fields in ***F/S tab*** and ***Roughing tab / Area Clearance tab*** of the Operation Parameters dialog box.

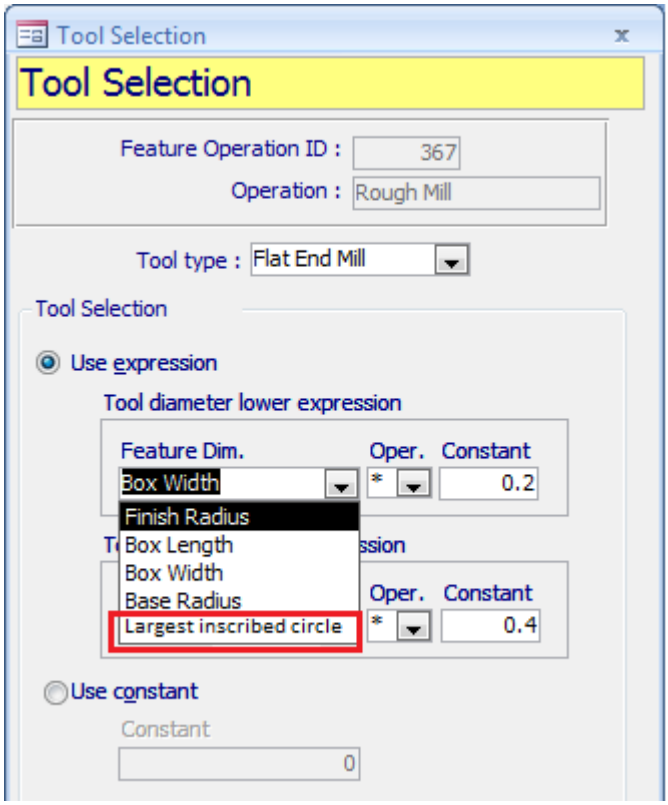
VoluMill Technology Expert Dialog Box

New - **Option to allow smooth repositioning of VoluMill toolpath moves**

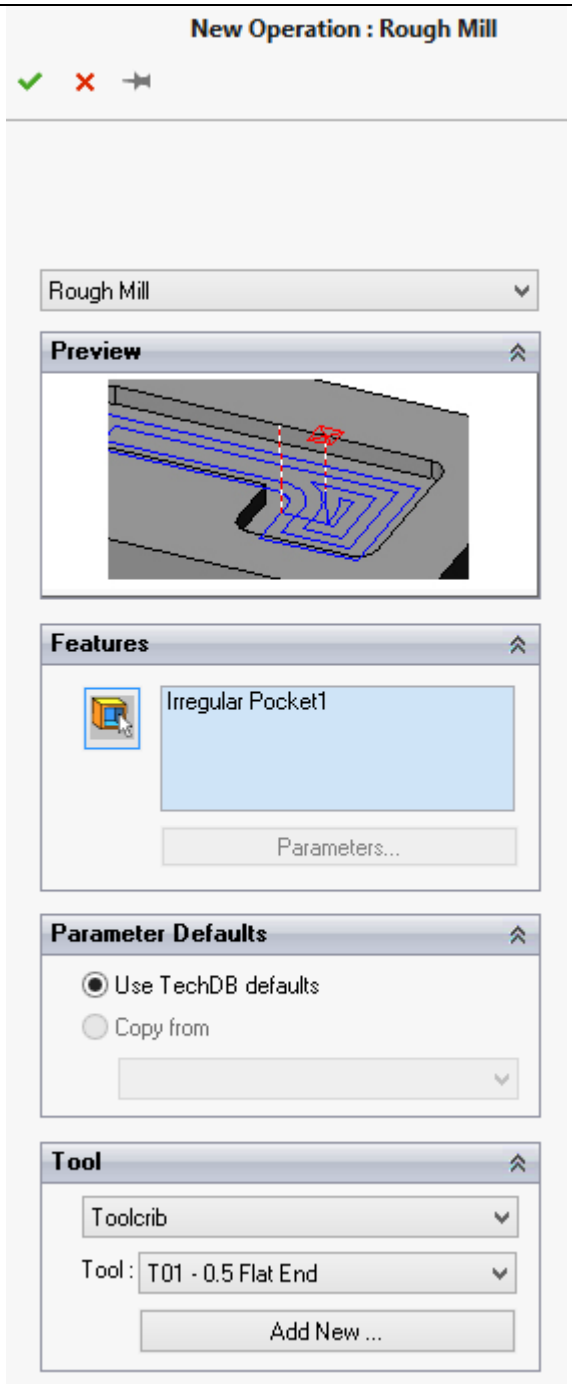
Purpose:	***Avoid sharp corners for repositioning*** checkbox option in *VoluMill Settings dialog box* to enable smooth repositioning of VoluMill toolpath when tool is not in contact with material
Implementation:	VoluMill toolpaths use high speed and feed rates. At such high feed rates, avoiding drastic direction changes (sharp corners), is recommended. When the tool is in contact with the material, you can define a ***Smoothing radius*** (as percentage of the tool diameter) in the ***VoluMill Settings dialog box***. This radius will be added to the toolpath to avoid drastic direction changes. When the tool is not in contact with the material, if the VoluMill toolpath makes sharp corners while repositioning, it may result in undesirable movements on the machine tool. This can be avoided by using the ***Avoid sharp corners for repositioning*** option in the ***VoluMill Settings dialog box***. - When this checkbox option is checked, the VoluMill toolpath will be smoothened using the internally set default values. - When this checkbox option is not checked, the VoluMill toolpath will be computed using sharp corners.
	 **VoluMill Settings Dialog Box**

New - Tool picking method based on the largest possible fitting tool

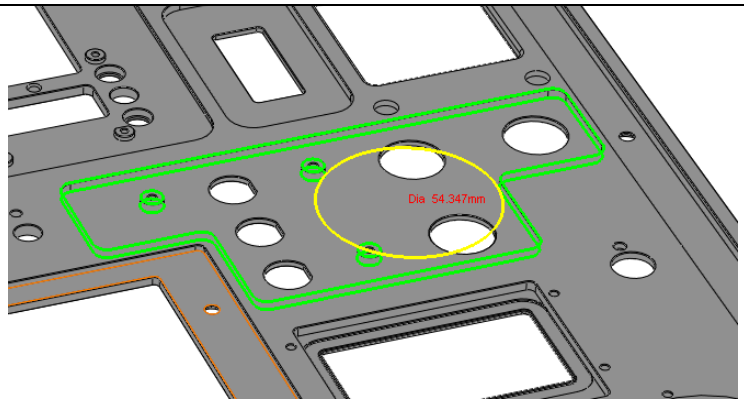
Purpose:	Provides the ***Tool*** selection method based on the largest fitting tool for closed profile features
Implementation:	Show the largest fitting circle in a closed shaped feature. The display is seen while inserting an operation and the tool option is set to From tool crib. The display is also seen in the graphics area when the tool tab of the operation parameter is active. Rules can be configured in TechDB as well for ***Largest inscribed circle*** option under the ***Tool Selection*** form.



Tool Selection Form



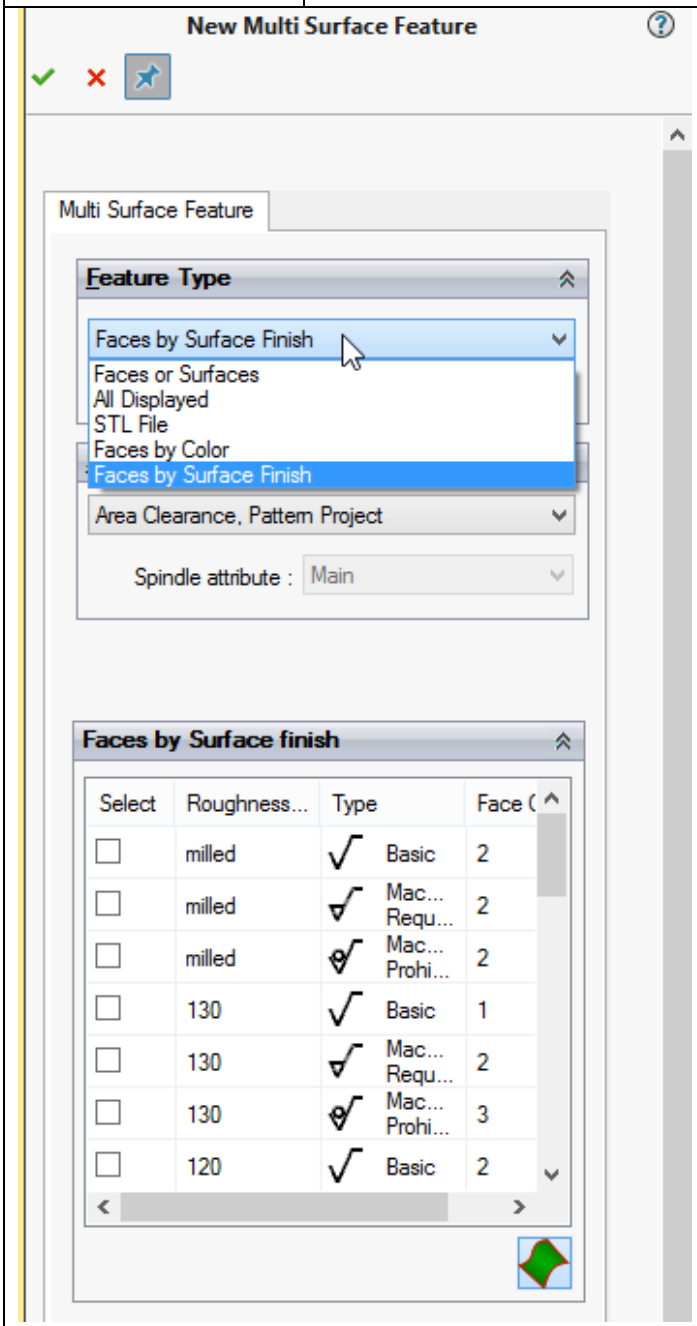
New Operation dialog box

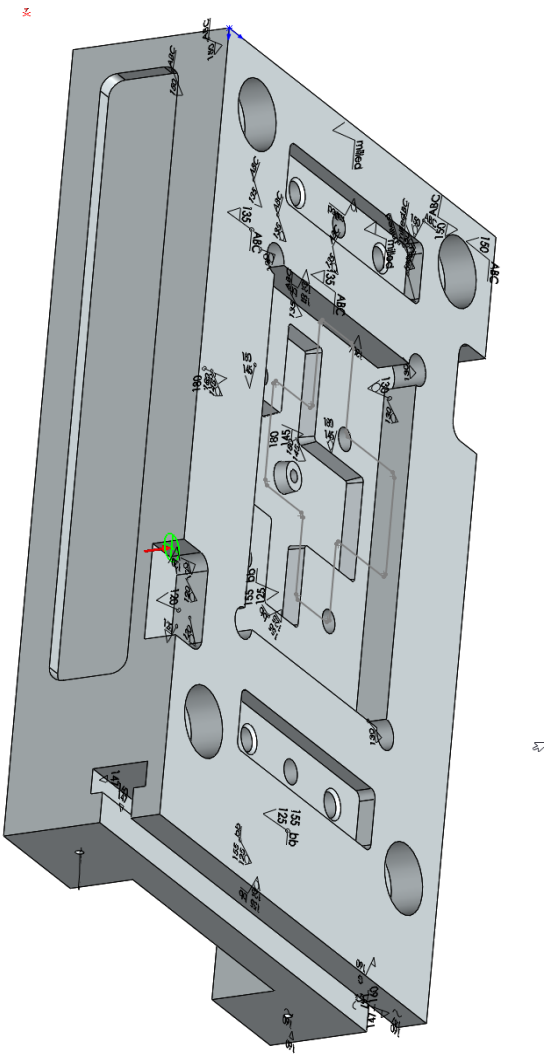


Tool selection based on Largest inscribed circle option

New - Defining Multi Surface Features from Surface Finish

Purpose:	Provide an option in the ***New Multi Surface Feature*** dialog box to define the multi surface feature based on Surface Finish type
Implementation:	In the ***Multi Surface Feature*** dialog box, when the ***Faces by Surface Finish*** option is selected, CAMWorks allows the multi surface feature to be defined from surfaces with specific types of surface finish and roughness values. The part faces will be grouped and listed in the ***Multi Surface Feature*** *dialog box* based on their Surface Finish Type and ***Minimum Roughness*** value. This method is useful when you have assigned surface finish properties and roughness values to the faces of the part in SOLIDWORKS.





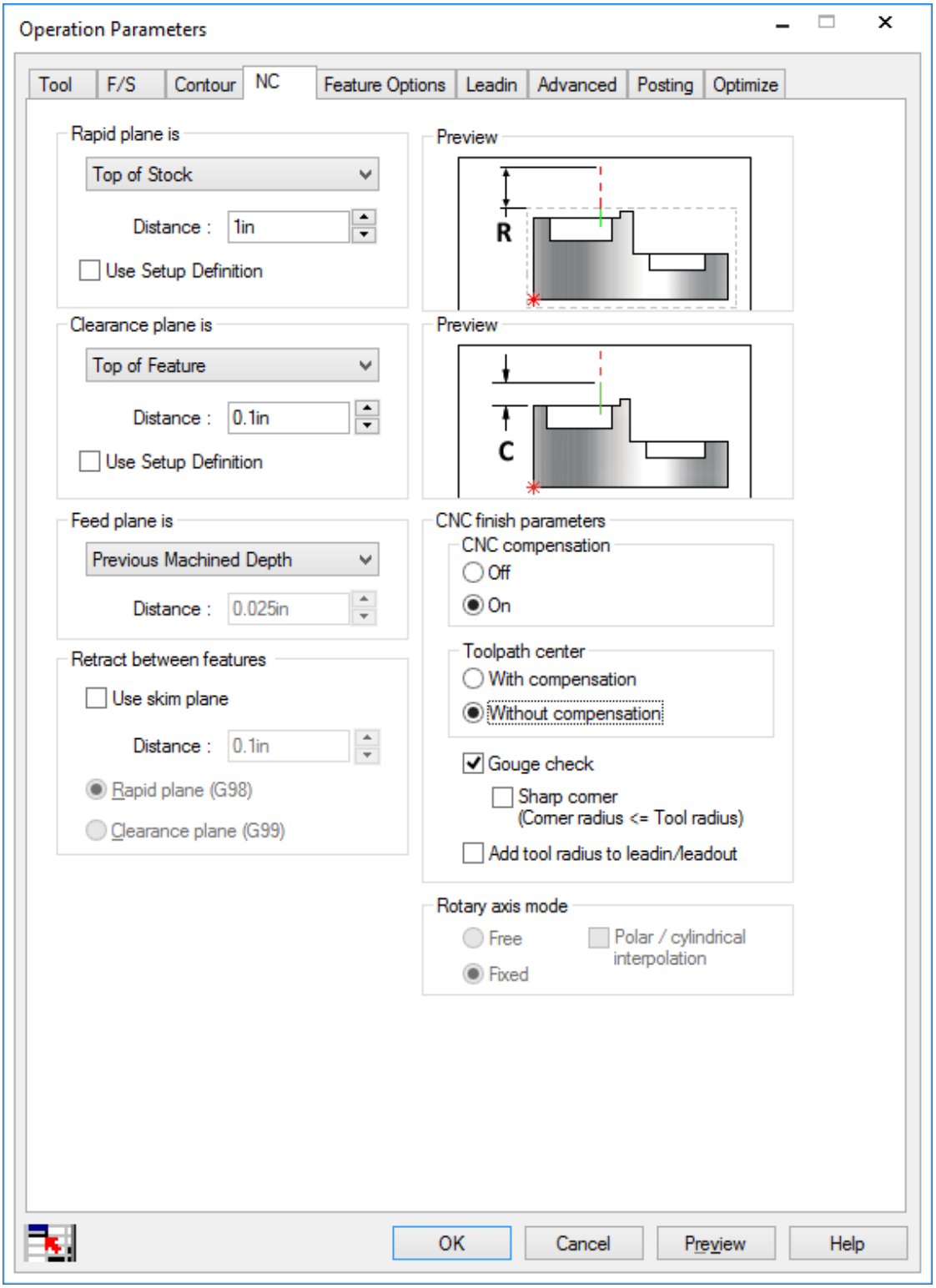
Define Multi Surface Feature dialog box

Improved - Improved CNC Compensation and Toolpath Center options for Contour Mill operations

Purpose:	Provide improved settings for CNC Compensation and Toolpath Center on the NC tab of Contour Mill operations.
Implementation:	**CNC comp options in Setup tab of Machine node shifted to NC tab of Contour Mill operation** The parameters in the ***CNC comp options*** group box of the ***Setup tab*** under Machine dialog box have either been removed or moved to the ***NC tab*** as follows: • The ***Display toolpath at G-code co-ordinates*** option has been removed. This option has been removed because the toolpaths will now always be displayed at the G-code co-ordinates. The ***Display cutter compensation on first move*** option has been removed. • The ***Calculate safe CNC toolpath*** option has been renamed to ***Gouge Check*** and moved to the NC tab of Contour Mill operation. • The ***Add tool radius to Leadin/Leadout*** option has been moved to the ***NC tab*** of Contour Mill operation. **How Gouge Check option in NC tab of Contour Mill operation affects Toolpath behavior** The ***Gouge Check*** option in the NC ab will be enabled when the ***CNC compensation*** is set to ***ON*** and ***Toolpath center*** is set to ***Without compensation***. When using CNC cutter compensation with full tool radius compensation, there can be conditions which can result in the part being gouged if the feature geometry contains areas that are narrower than the diameter of the tool. • When this option is checked, the toolpath will be computed by considering the entire feature geometry and eliminating portions of the toolpath that may result in gouging of the part. • When this option is not checked, the toolpath output into G-code is effectively the same as the feature geometry. If a feature contains areas that are smaller than the tool radius, it is possible that the feature may be gouged. **Newly introduced 'Sharp Corner (Corner radius<Tool radius)' option in NC tab** When ***Toolpath center*** is set to ***Without compensation***, the tool will traverse on the periphery of the feature. If the corner radius of the feature being machined is smaller than or equal to the tool radius, there are two possible ways to generate the toolpath: • Generate an arc move at the corner of the feature. The radius of the arc will be equal to the radius of the tool. OR • Generating a sharp corner Previous versions of CAMWorks didn't provide any method to choose the above-mentioned ways to generate the toolpath. From *CAMWorks 2017* version onwards, this functionality is provided in the form of the ***Sharp Corner (Corner radius<Tool radius)*** option. This checkbox option will be enabled only when the ***Gouge check*** option is checked else it will remain disabled. • When the ***Sharp corner*** checkbox option is unchecked, the toolpath machining the feature will maintain an arc at the corners of the feature. The radius of this arc will be equal to the radius of the tool used for

machining the feature.

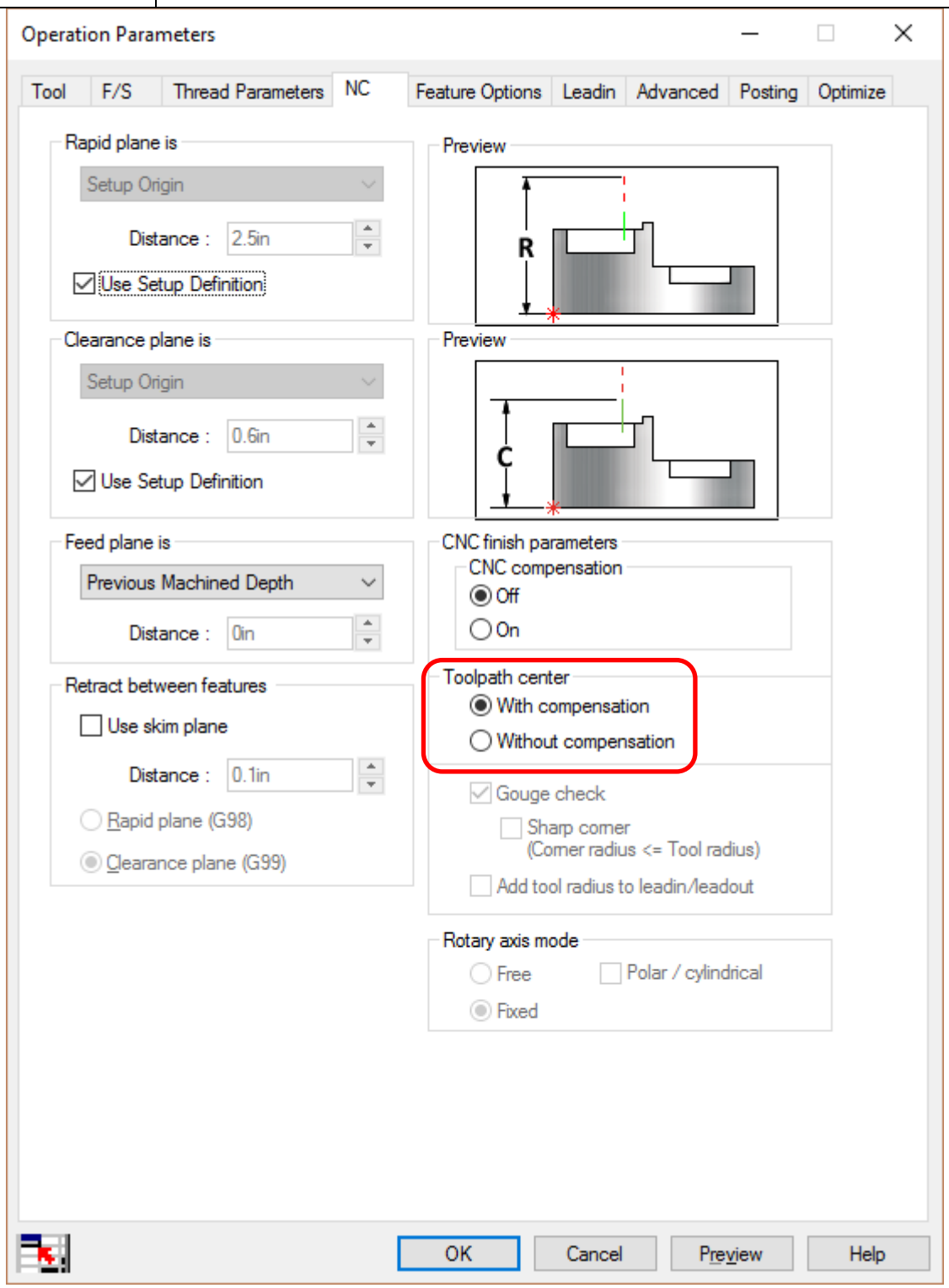
- When the ***Sharp Corner*** checkbox option is checked, the toolpath will maintain a sharp corner for the feature provided all below conditions are fulfilled:
    - ***CNC compensation*** is set to ***ON***
    - ***Toolpath center*** is set to ***Without compensation***
    - ***Gouge Check*** option is checked
    - Corner radius of the feature being machined is lesser than or equal to the radius of the toolpath



NC tab of Contour Mill operation

New - Allow Toolpath Center Options for Thread Milling

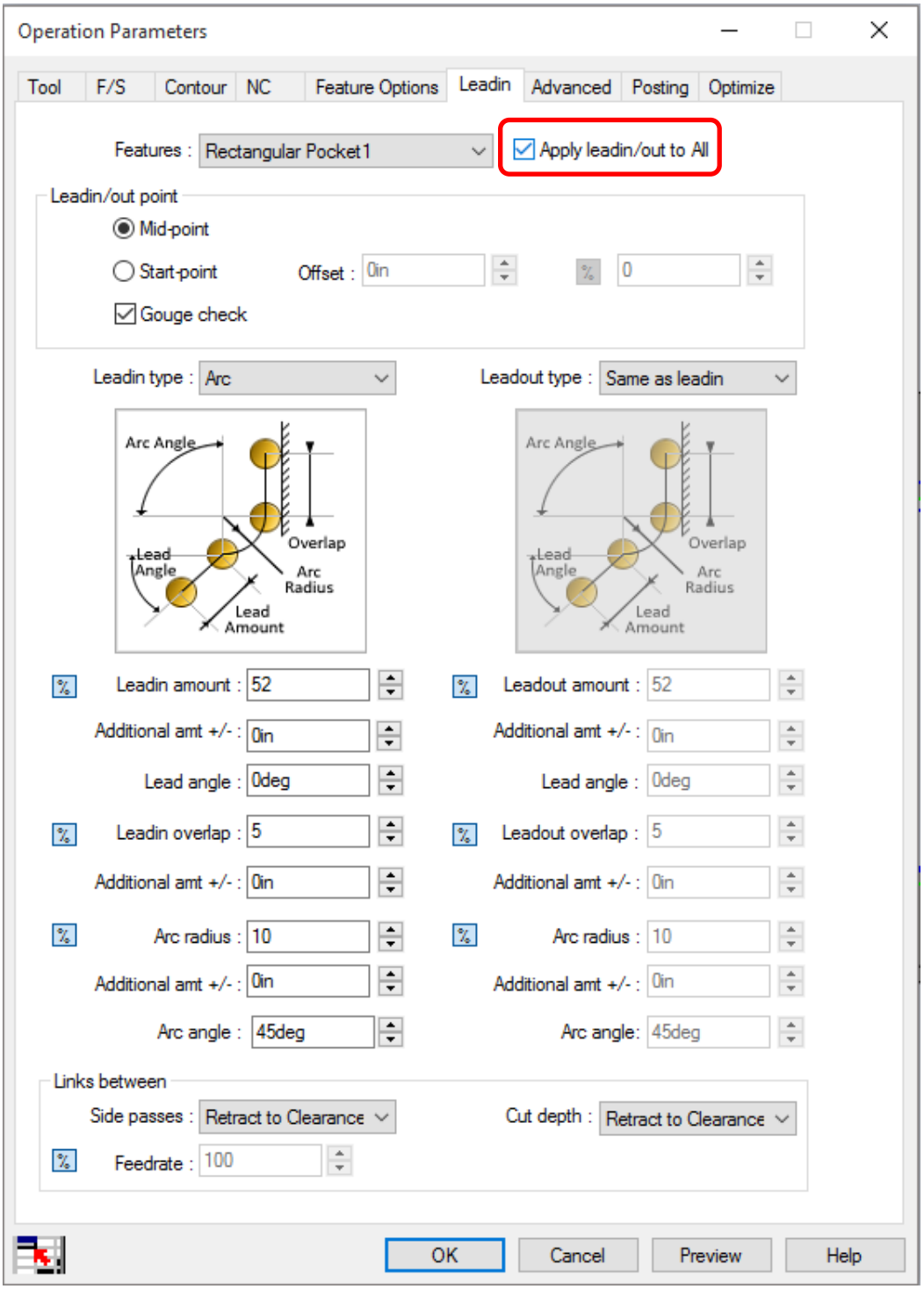
Purpose:	Allows you to use the options of Toolpath center under CNC finish parameters for thread milling operations
Implementation:	From ***CAMWorks 2017*** version onwards, you can use the options of toolpath center i.e. with compensation or without compensation for thread mill operations. The pitch defined by the user is maintained in both the options.



Toolpath Center Options for Thread Milling

Modified - 'Apply Leadin/out to All' option for Contour Mill operations

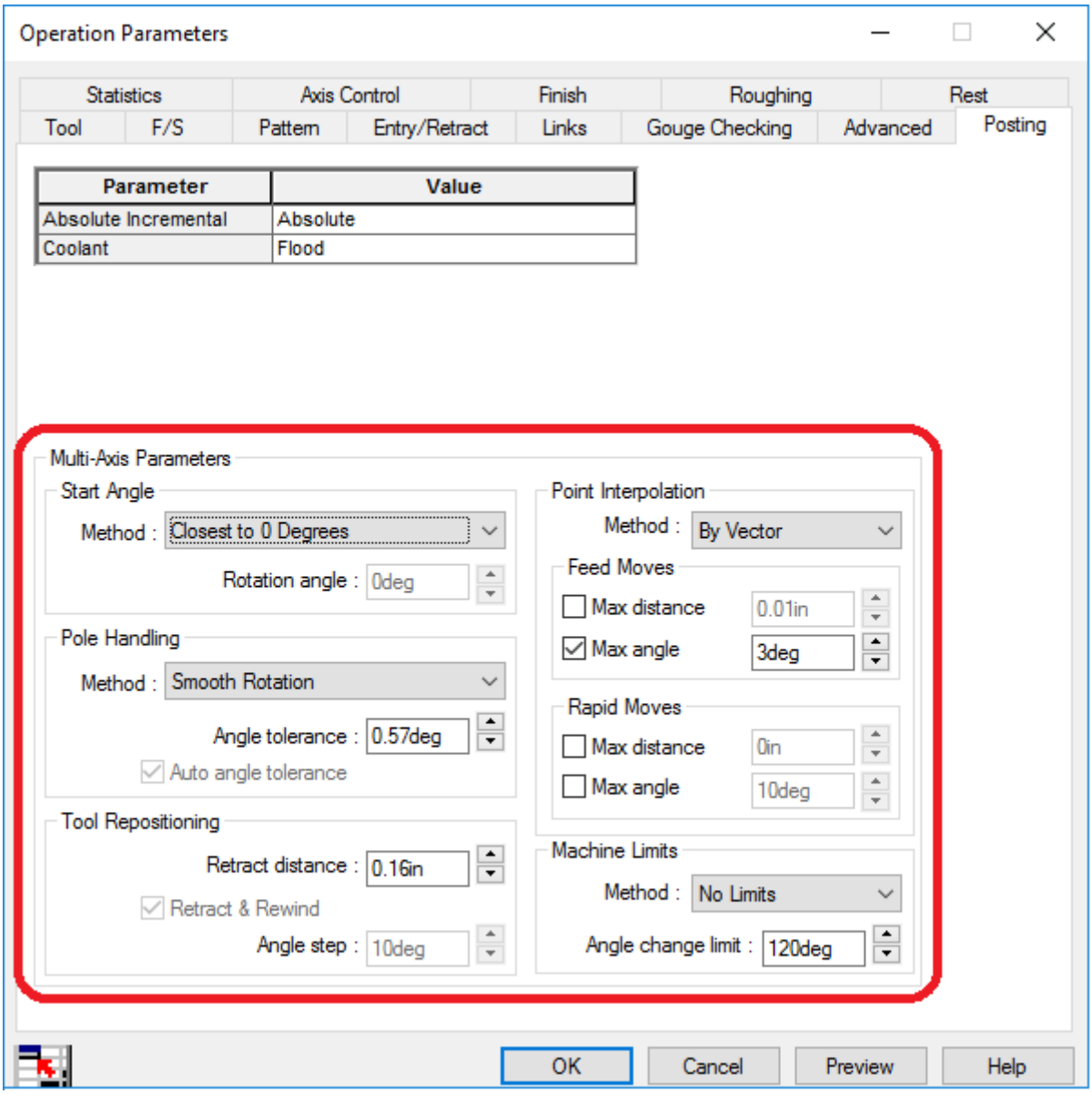
Purpose:	To provide modified User Interface within CAMWorks for ***Apply Leadin/out to All*** in Contour Mill operations for easier identification
Implementation:	The push button for ***Apply to Leadin/out to All*** on the ***Leadin*** tab of Contour Mill operation has been changed to a check box. This helps in easier identification of the status of the control.



Apply Leadin/out to All option in Leadin Tab of a Contour Mill Operation

Improved - Support for Multiaxis Post Parameters in Operation Parameters dialog box

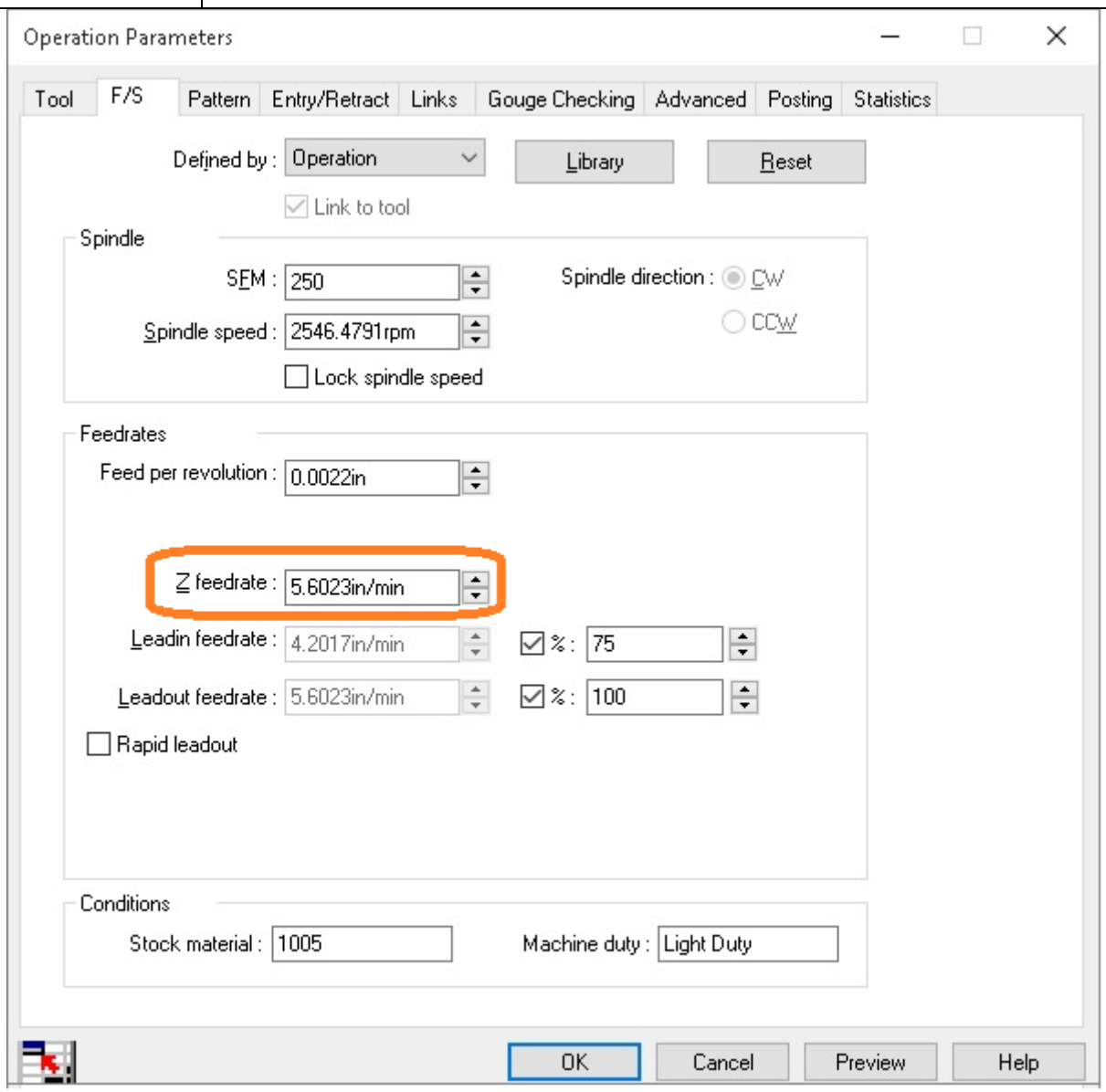
Purpose:	Provide a user-friendly interface to users for selecting and entering parametric values associated with post processor within the CAMWorks User Interface
Implementation:	From *CAMWorks 2017* version onwards, parameters associated with multiaxis post will be located on the ***Posting*** tab of the 3 Axis and Multi axis Mill operations. These parameters will only be active if the post processor selected in the Machine definition supports the parameters; else all these parameters will be disabled. They are supported for all multiaxis mill operations and 3 Axis Mill operations except the following operations: • Area Clearance • Legacy Rough Mill • Legacy Contour Mill



Multiaxis parameters in the Posting Tab of Mill Operation

Improved - Support for Z feedrate in Multi Axis Drilling and 5 Axis Plunge Roughing

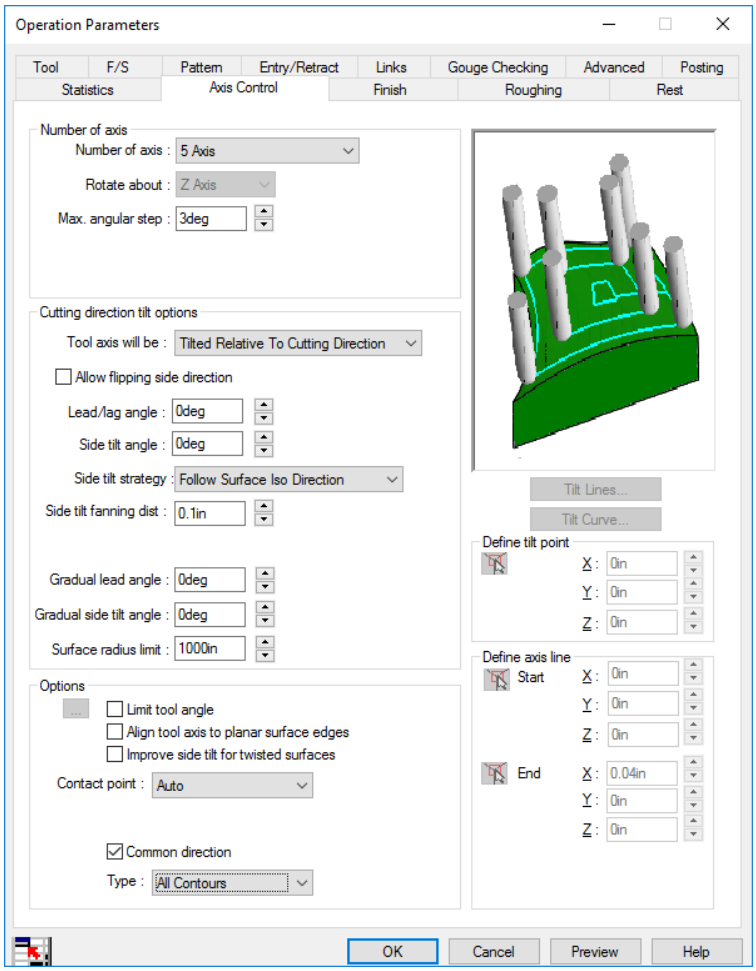
Purpose:	All Multiaxis drilling operations and Plunge Roughing operations will now use Z feedrate and not XY Feedrate as feedrate computation for Drilling is different from Milling operations.
Implementation:	The Z Feedrate parameter in the F/S tab of a drilling operation indicates the feedrate for any Z tool cutting movement. Feedrates are given in inches per minute or millimeters per minute. ▪ For 2.5 Axis Mill operations, you can specify the Z feed rate explicitly or as a percent of the XY feed rate in the F/S tab. ▪ For all single point operations, ***Multiaxis drilling*** operations and ***Multiaxis Plunge Roughing*** operations, you can specify the Z feedrate explicitly. **Note:** In previous versions of CAMWorks, XY Feedrate was used for Multiaxis drilling operation and Multiaxis Plunge Roughing operations for computation of Feeds and Speeds. This has now been corrected to use Z Feedrate.



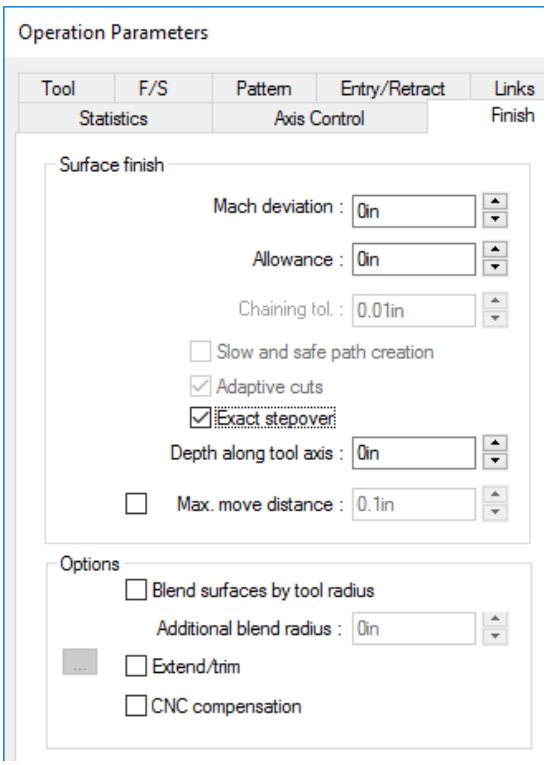
Z Feedrate parameter in F/S tab of Operation Parameters Dialog Box

New - Options for Axis Control and Surface Finish for Multiaxis Mill operations

Purpose:	Provide options in the ***Axis Control*** and ***Finish*** tabs of Operation Parameters dialog box for achieving better surface finish
Implementation:	New parameters have been added in the ***Axis Control*** and ***Finish*** tabs of ***Operation Parameters*** dialog box that allow users to: • Define tilt direction based on contours • Optimize the tilting of tool during cutting • Define exact step over to be achieved during machining between flow lines



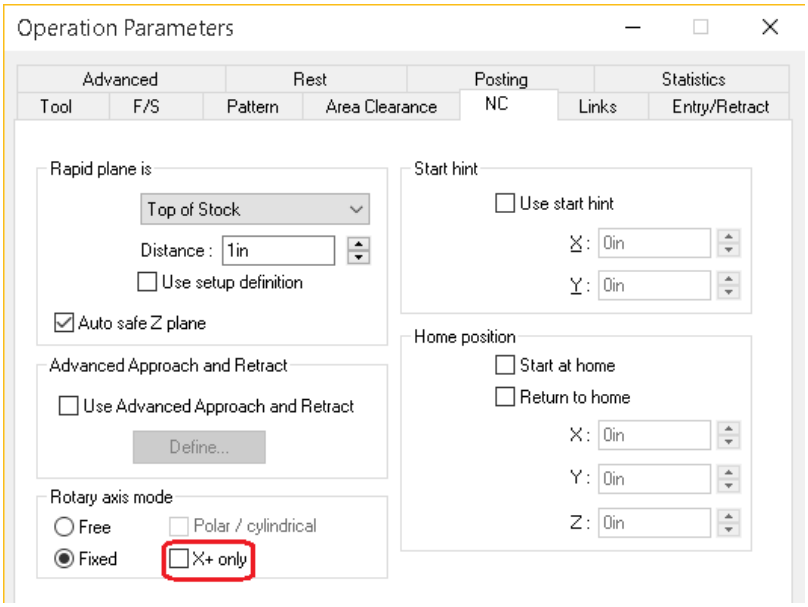
Axis Control tab

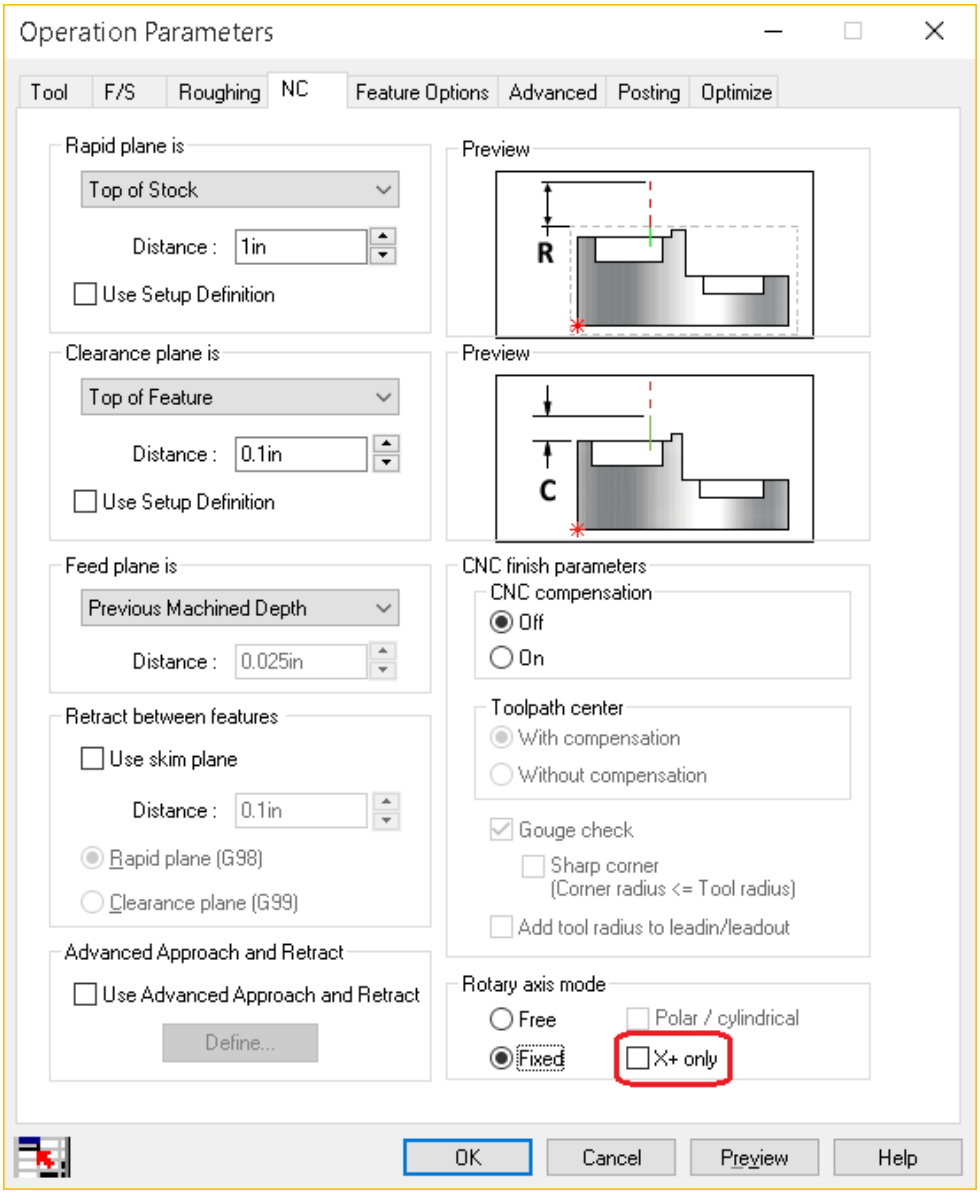


Finish tab

Turn/Mill-Turn

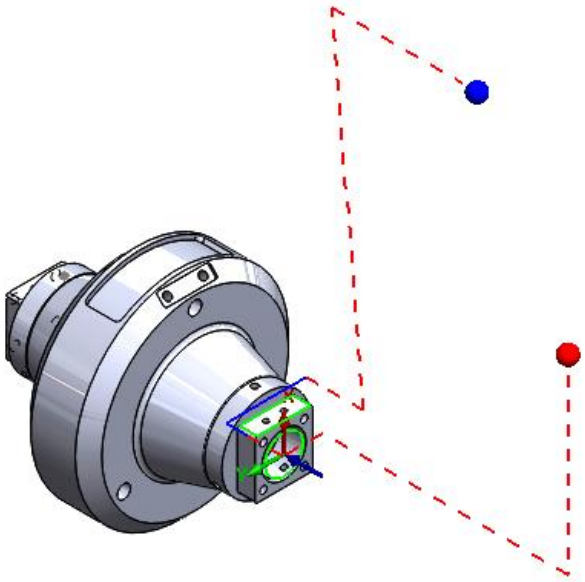
New - X-Plus only support for Mill Operations on the Face Setup

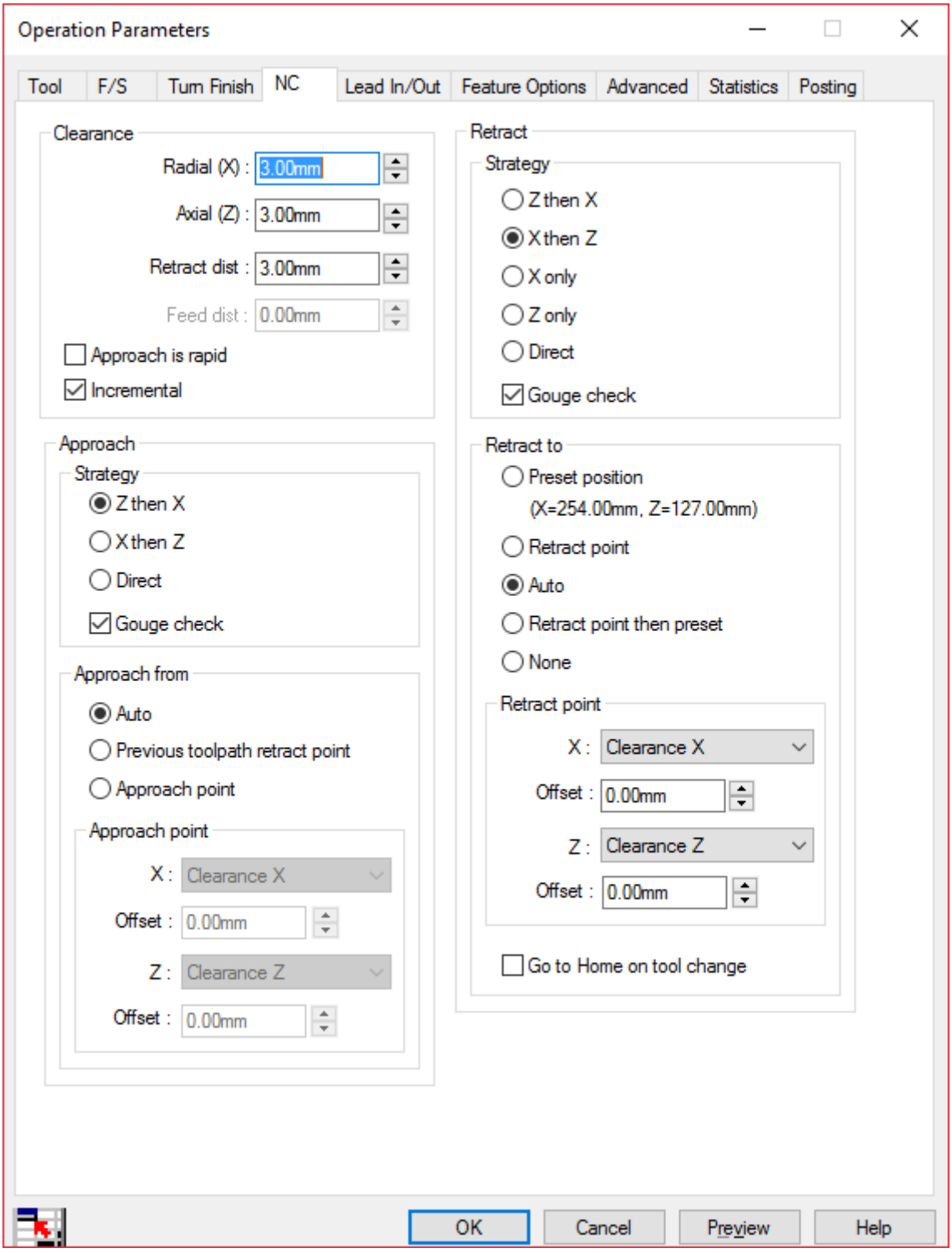
Purpose:	Provide an option whereby posted NC code for Face Milling Operations of Mill-Turn will not have any negative X co-ordinates.
Implementation:	Some Mill-Turn machines have very limited or no movement in the negative X direction. Under such conditions, if you have to machine a feature on the face of the part, then the movement of the spindle has to be controlled in such a way that the posted code will have moves only in the positive X direction. For this purpose, the following checkbox option has been introduced in *CAMWorks 2017*: • ***X+ only*** checkbox option in *Rotary axis mode* group box on ***NC tab*** of 2.5 Axis operations. • ***X+ only*** checkbox option in *Rotary axis mode* group box on ***Axis tab*** of Multiaxis operations. **How this 'X+ only' checkbox option works:** • For 2.5 Axis operations, the ***X+ only*** checkbox option is enabled only when the *Rotary axis mode* is set to ***Fixed***. • For multiaxis operations, the ***X+ only*** checkbox option is enabled only when the *Rotary axis mode* is set to ***Fixed*** and the ***Number of axis*** is set to 3. When this checkbox option is checked, the posted NC code for the operation will not have any negative X co-ordinates. Instead, the posted NC code will be modified such that C Axis co-ordinates will be automatically added to ensure that the tool cuts only positive X direction to machine the target feature. **Example:** Consider that the tool has to move from *X+10, Y0* position to *X-10,Y0* position. When the ***X+ only*** checkbox option is checked, then this move will be split into two moves. First, a move of *C90* will be added. Subsequently, the second move will be from *X0,Y-10* to *X0,Y+10* position. The tool will thus make a vertical move instead of a horizontal move.  **Axis tab on Multiaxis Mill operation**



X+only checkbox option in NC Tab of a 2.5 Axis Mill operation

Improved - Enhanced Approach and Retract to define the points with respect to stock/ clearance/origin

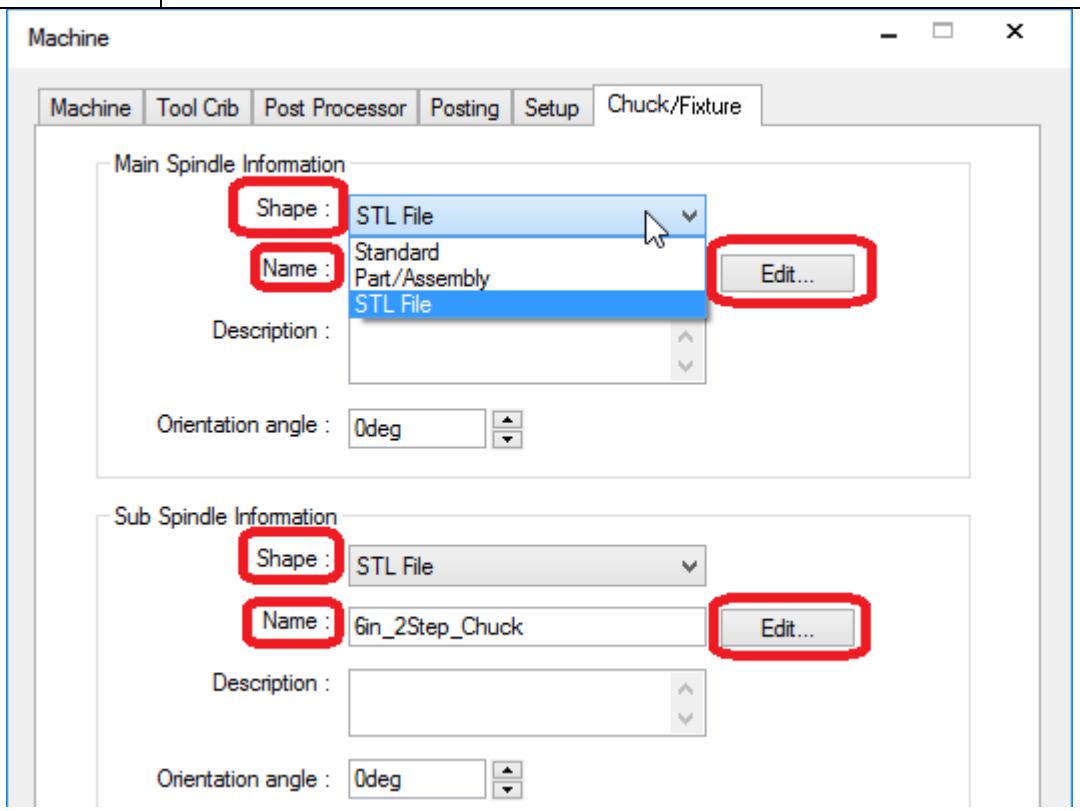
Purpose:	Provide options in the NC tab of a Turn operation that enable users to define Approach and Retract positions of the tool with respect to the Stock or Work in Process.
Implementation:	Use the various parameters in this tab to define the Approach and Retract strategies and positions. **Display of Approach and Retract Points in Graphics area** When the ***NC*** tab of a turn operation is active, the approach and retract points for the tool will be displayed in the Graphics area. By default, the Approach point is displayed in blue color and Retract point in red color.  **New retract strategies of X only and Z only** The new strategies of X only and Z only allow the user to retract the tool in either of the axis. This method helps while machining lengthy parts. **New retract position of Auto** The option of ***Auto*** in ***Retract to*** group box positions the tool at the preset position in case of a tool change. Else the next operation using the same tool will start from the last point of the current operation. **New approach options of Auto and Approach Point** The *Approach from* options of ***Auto, Previous toolpath retract point*** and ***Approach point*** assist the user in selecting an approach point in case needed. Else the automatically decided approach point based on the start point of the current toolpath and retract point of the preceding toolpath will be considered.



NC Tab of Turn Operations

New - Define Custom Chuck/Fixture shape using STL File or solid part/assembly

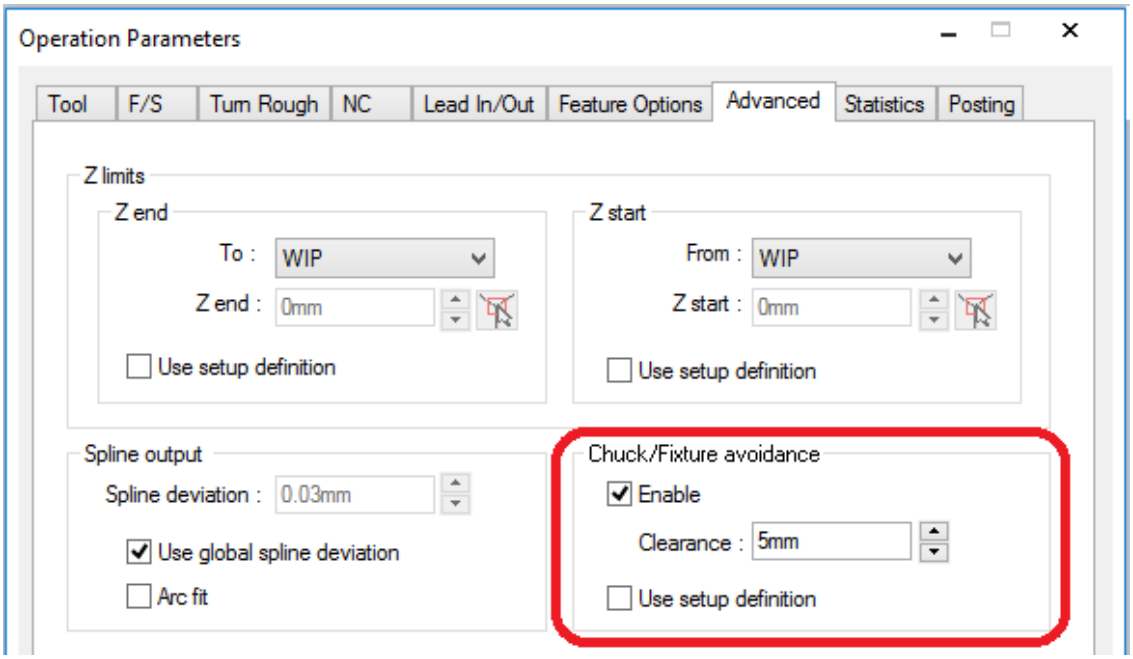
Purpose:	Provides ability to define chuck/fixture using STL Files or part/assembly files
Implementation:	The chuck/fixture shape for Main Spindle and Sub Spindle is defined in the ***Chuck/ Fixture*** tab of the ***Machine Definition*** dialog box. In previous versions of CAMWorks, chuck shape could be defined only by using/editing/creating a Chuck Configuration. From ***CAMWorks 2017*** version onwards, the chuck shape can optionally be defined from a STL File or a part/assembly file. This functionality is available in the form of the ***Shape*** dropdown list in the *Chuck/Fixture* tab. • Select ***Standard*** in this dropdown list to define chuck using a Chuck Configuration • Select ***STL File*** in this dropdown list to define chuck using an STL File • Select ***Part/Assembly*** to define chuck using a solid part or assembly. Clicking on the ***Browse*** button  next to the ***Shape*** dropdown list displays the Windows Explorer ***Open*** dialog box. Use this dialog box to select the desired STL/part/assembly file that defines the shape of the chuck. Clicking on the ***OK*** button of this dialog box confirms the selection. The name of the selected file will then be displayed in the ***Name*** field within the ***Chuck/Fixture*** tab and the file representing the chuck can be viewed in the graphics area. Clicking on the ***Edit*** button opens the ***Define Chuck*** dialog box. Use this dialog box to move or rotate the chuck in X, Y and Z axis direction with respect to the Fixture Coordinate System (FCS) of the current part so that the chuck is aligned with this current part.



Selecting chuck Shape in Chuck/Fixture tab of Machine dialog box

New - Automatic Chuck detection for Turn Toolpath Avoidance

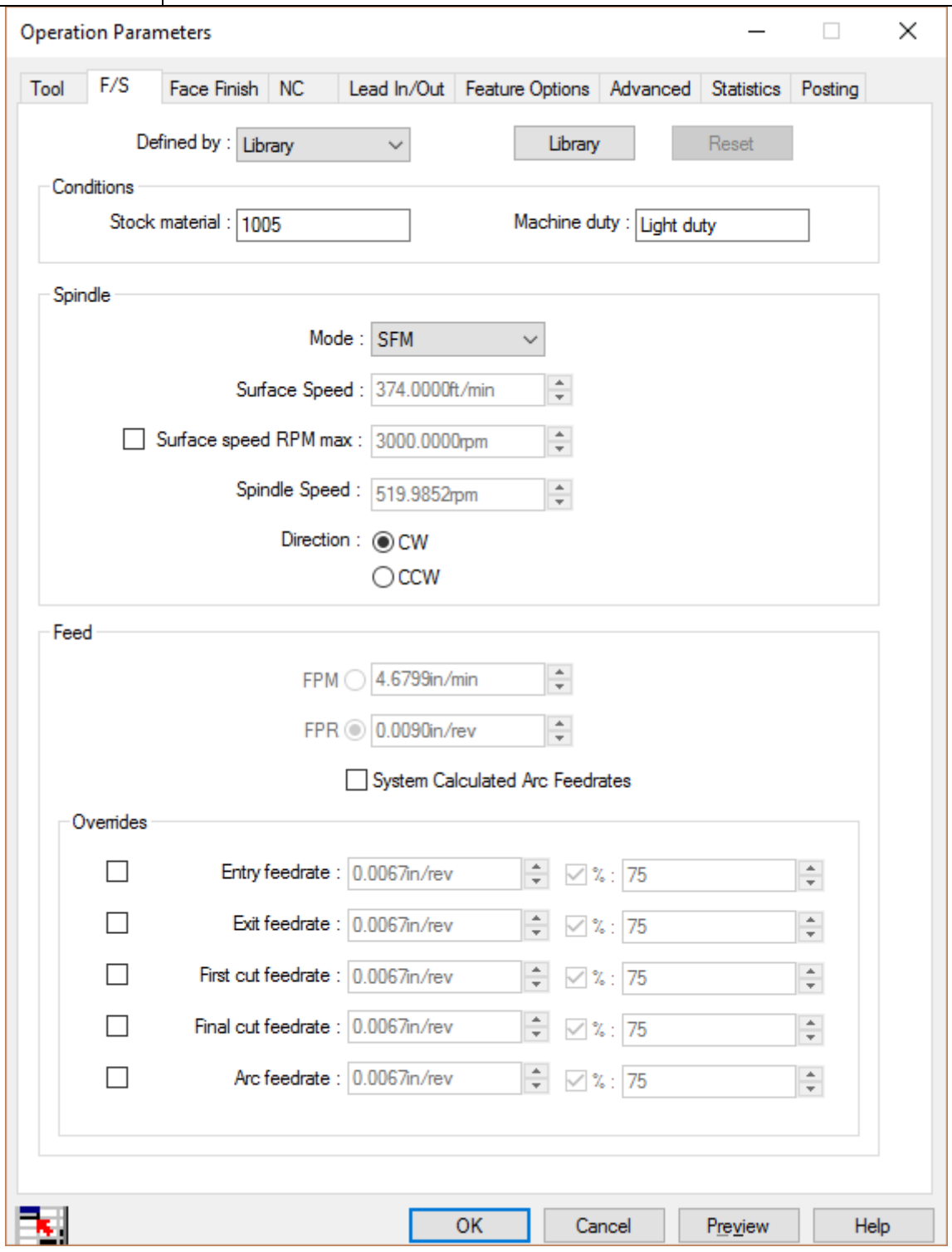
Purpose:	For Turn and Mill-Turn parts, turn toolpaths will now be generated after considering the Chuck geometry and chuck location.
Implementation:	**Chuck/Fixture Avoidance option in Advanced tab of Operation Setup Parameters dialog box & Operation Parameters dialog box** When the *Chuck/Fixture Avoidance* functionality in the ***Advanced*** tab of *Operation Setup Parameters* dialog box is enabled and applied to all the operations in the setup then, the toolpaths will be generated after considering the location of the chuck (thereby avoiding possible collisions). Collision with chuck/fixture is avoided by defining a *Clearance* value for the toolpath to stay away from the chuck/fixture. The Chuck/Fixture avoidance settings defined in this tab are global in nature. To override the global value or in case no global value is defined for a specific turn operation, uncheck the ***Use Setup Definition*** option for Chuck/Fixture avoidance in *Advanced* tab of Turn Operation Parameters dialog box and set desired *Clearance* value. **Note:** Default settings for Chuck/Fixture Avoidance can be set in *Advanced* tab of Turn Operation Parameters Form in the Technology Database. **Graphical display of Toolpath Limit when Chuck/Fixture Avoidance is enabled** When Chuck/Fixture avoidance is enabled in this tab, a graphical representation of the position of chuck/fixture avoidance will be displayed in the graphics area. The graphical representation consists of lines indicating the toolpath limits based on the Clearance value for Chuck/Fixture avoidance defined in this tab. Toolpath limits for both ID and OD toolpaths will be displayed in the graphics area. If ***Chuck/Fixture avoidance*** is enabled in this tab, these toolpath limits will be displayed in the graphics area in the following cases: ▪ When the Turn Setup node is highlighted in the CAMWorks Operation tree ▪ When the ***Operation Setup Parameters*** dialog box for a Turn Setup in the Operation tree is open and the Advanced tab is the active tab.



Chuck/Fixture avoidance funcionality in Operation Parameters dialog box

New - Feed/Speed tab for Turn Operations

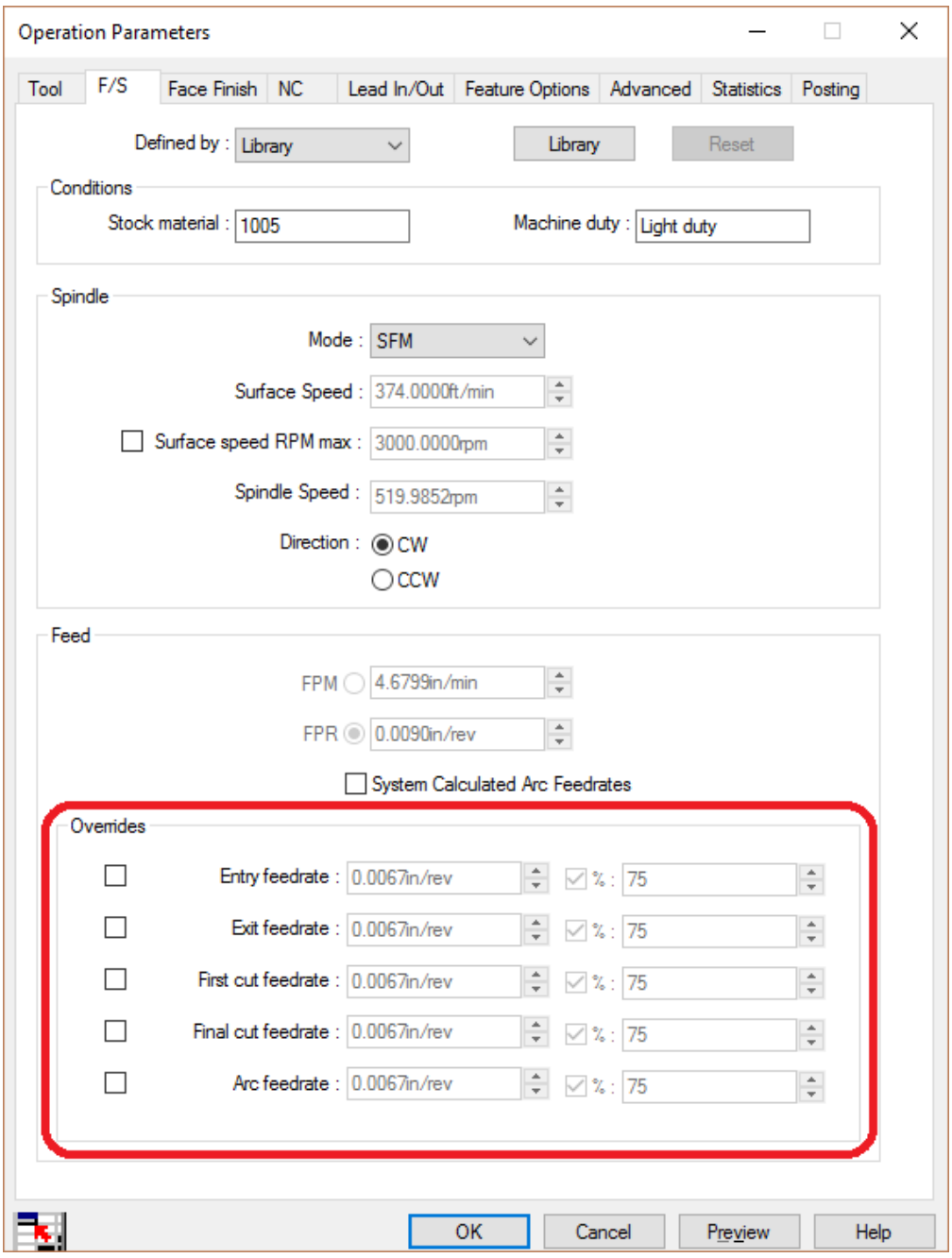
Purpose:	Creation of separate ***F/S*** tab for Turn operations and provide all new and existing parameters related to Feed and Speed under this tab.
Implementation:	In the earlier version of CAMWorks, there was no separate F/S tab for turn operations. From *CAMWorks 2017* version onwards, the existing F/S parameters have been moved from NC tab to newly created F/S tab. The ***Stock material*** and ***Machine duty*** parameters in this tab indicate the corresponding info that will be passed on to the Feed/Speed editor.



F/S tab of Turn Operations

New - Feedrate Overrides for Turn Operations

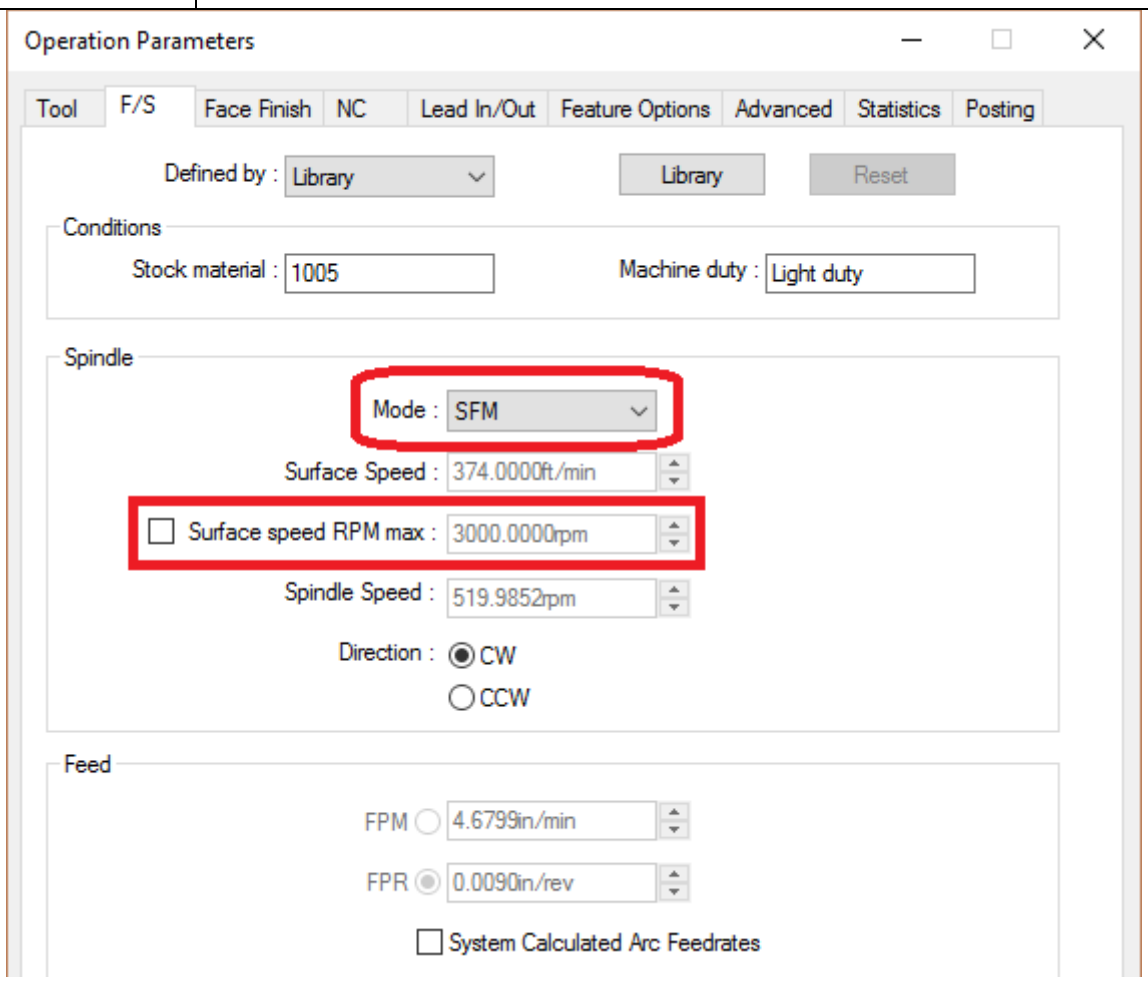
Purpose:	Allow you to assign different feed rates for entry moves, first cut and/or last cut moves and at corners having fillets.
Implementation:	On the Feed/Speed tab of all Turn operations except Threading, new parameters to override the cutting feedrate have been provided. These parameters include the following: • Overriding feedrate for entry moves(Leadin/Step in) • Exit moves(Leadout/Step out) • First cut moves • Last cut moves You can also override the feedrates at corners having concave and convex fillets. You can also set the option of automatically calculating the feedrates at the corners having arcs.



F/S tab in Turn Operation Parameters dialog box

New - Allow Max. Spindle Speed per operation

Purpose:	To have a control to set the maximum spindle speed for every turn operation
Implementation:	A new check box has been added to the F/S tab of Turn operations to set the maximum possible RPM for the current operation. The RPM for the operation will be set to the maximum allowed as set by you in case the calculated RPM exceeds the limit. The check box will be active when ***Spindle mode*** is set to ***SFM***. A new variable has also been added so that the post processed G-code will have the maximum RPM set by the user.

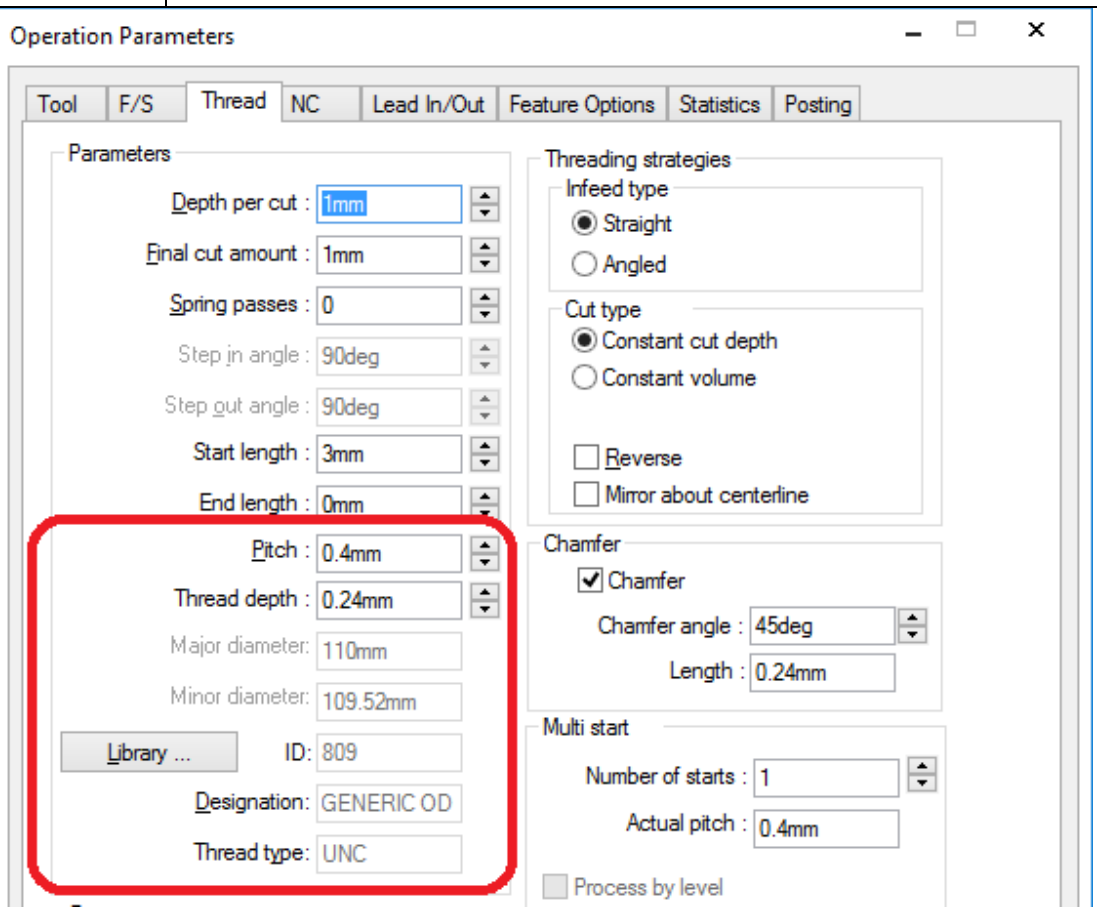


F/S tab in Turn Operation Parameters dialog box

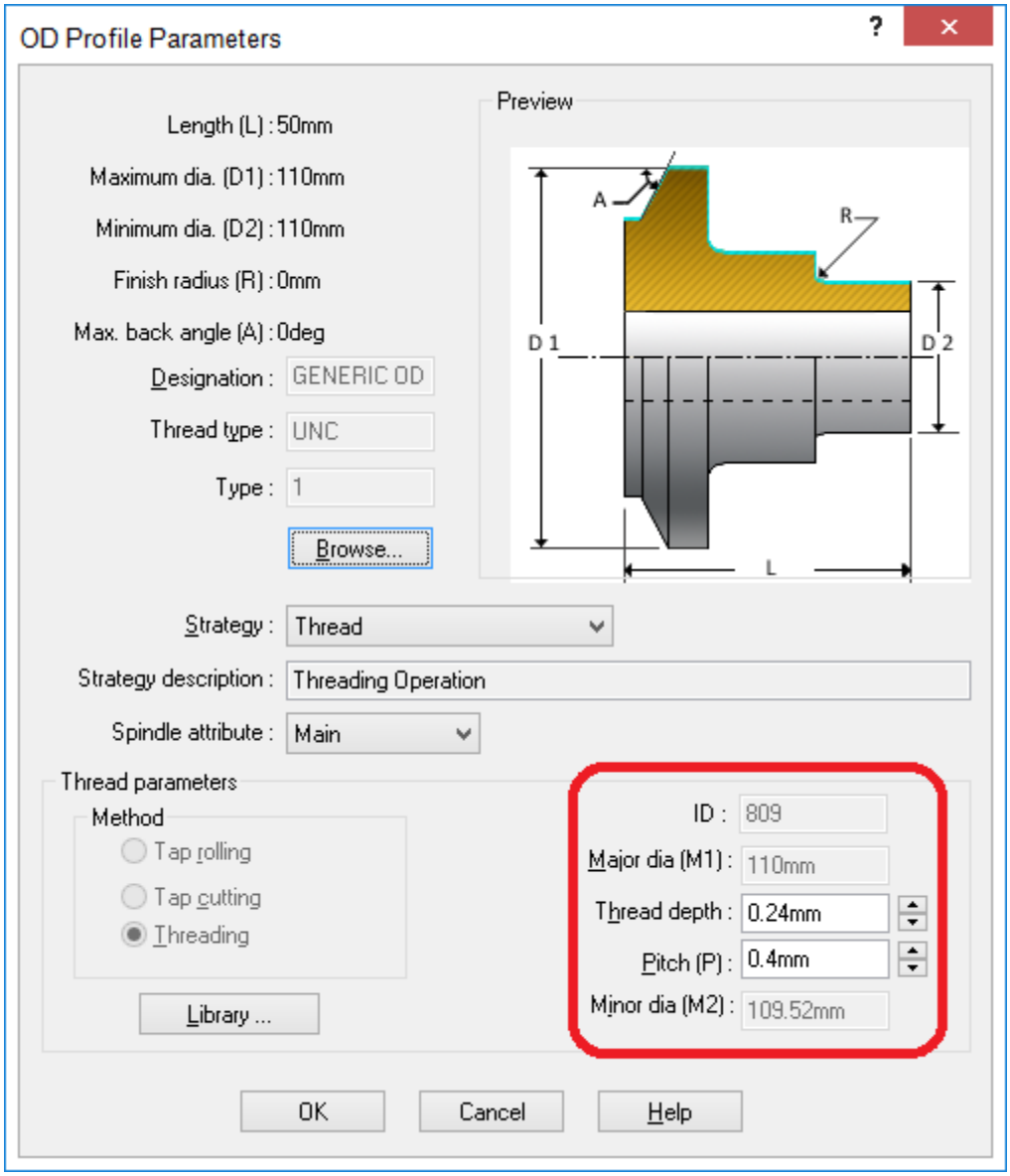
Improved - Turn Thread Operation User Interface

Purpose:	Improved user interface of turn threading operation where the user can select or modify the thread condition within the operation. Real time values of the features are populated in the designated fields.

Implementation:	*CAMWorks 2017* will have the following new functionalities for Turn Thread operations which will be available in the ***Thread*** tab of the Turn Thread operation:
	• Ability to edit ***Thread Depth*** parameter
	  ○ For an ID feature, the *Major Diameter* will be calculated/updated as the sum of the *Minor diameter* and twice the thread depth.
	  ○ For an OD feature, the *Minor Diameter* will be calculated/updated as the difference between the *Major diameter* & twice the thread depth.
	• Major and Minor diameters defined the ***OD Feature Parameters*** or ***ID Feature Parameters*** dialog box will be populated in corresponding fields in the ***Thread*** tab of the operation
	• ***Library*** button to open the TechDB and select a desired Thread Condition record.
	• Read-only ***ID*** field displaying the TechDB ID of the ***Thread Condition record*** selected for the Thread feature.
	• ***Designation*** field displaying the description of the selected Thread condition from TechDB.
	• ***Thread Type*** field displaying the associated thread type for the selected thread definition.
	Note: If any parameter (*Turn Depth, Major Diameter, Minor Diameter, Thread Condition record*) in the ***Thread*** tab is updated, then the corresponding value in the ***OD Profile Parameters*** or ***ID Profile Parameters*** dialog box too will be updated.



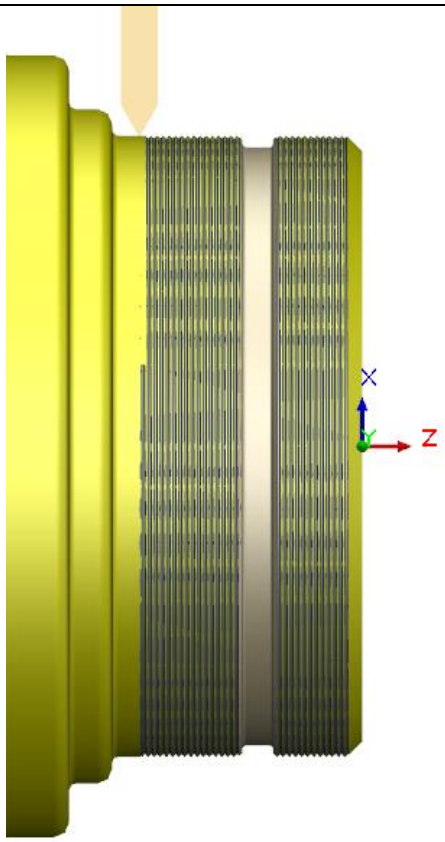
Turn Thread tab in Operation Parameters Dialog Box



Thread Parameters in OD Profile Parameters Dialog Box

New - Defining Turn Thread Feature using multiple collinear segments

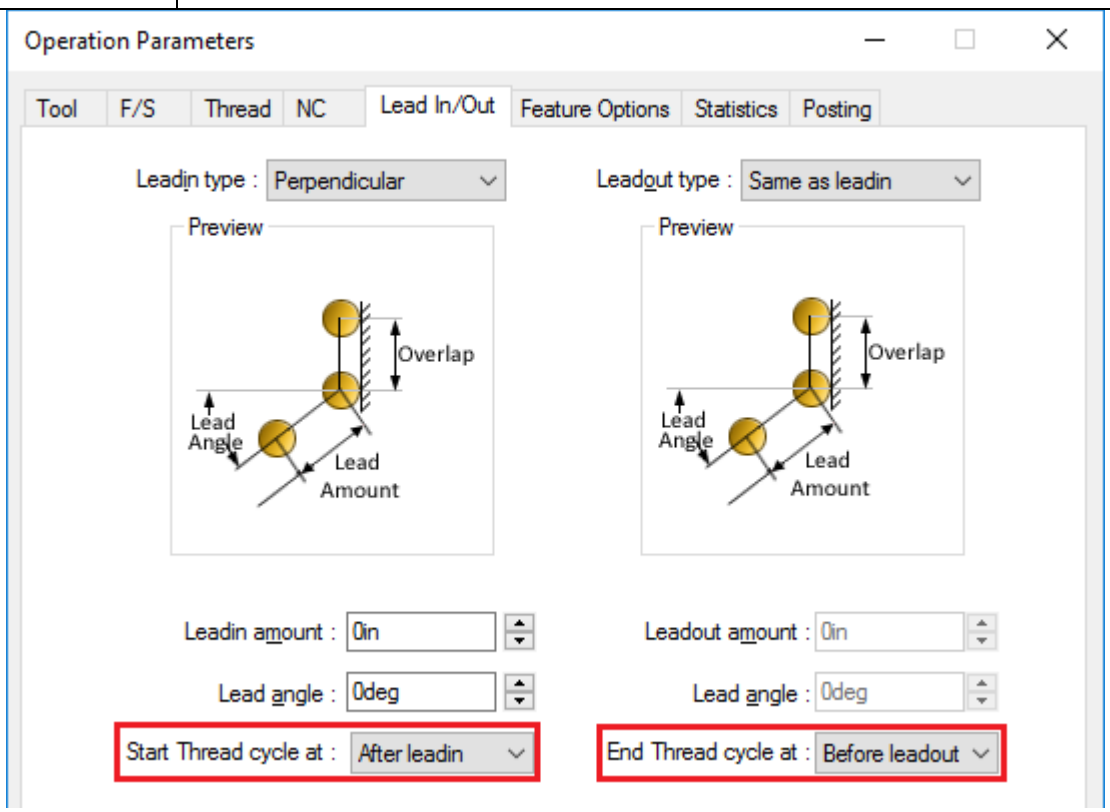
Purpose:	Support generation of Turn Thread toolpath when multiple collinear line segments are selected in the Turn Thread feature definition
Implementation:	In previous versions of CAMWorks, Turn Thread features could be interactively defined using only singular line segments. From *CAMWorks 2017* version onwards, Turn Thread feature definition supports selection of multiple collinear line segments (i.e. the selected segment entities must lie in a straight line) for defining thread feature. Toolpaths can be generated for such Turn Thread features. This functionality will be supported for Turn Thread for feature types of Face, OD and ID. Even disjoint segment entities that are not directly connected but are collinear in nature can be used to define a thread feature by defining a straight join segment. **Note:** On inserting a thread feature, when its constituent segment entities are not collinear in nature, an error message will be displayed indicating that the feature is invalid for the thread operation and hence no toolpath would be generated.



Thread feature with Disjoint segment entities that are not directly connected but are collinear in nature

Improved - Option for Start and End of Turn Thread cycles

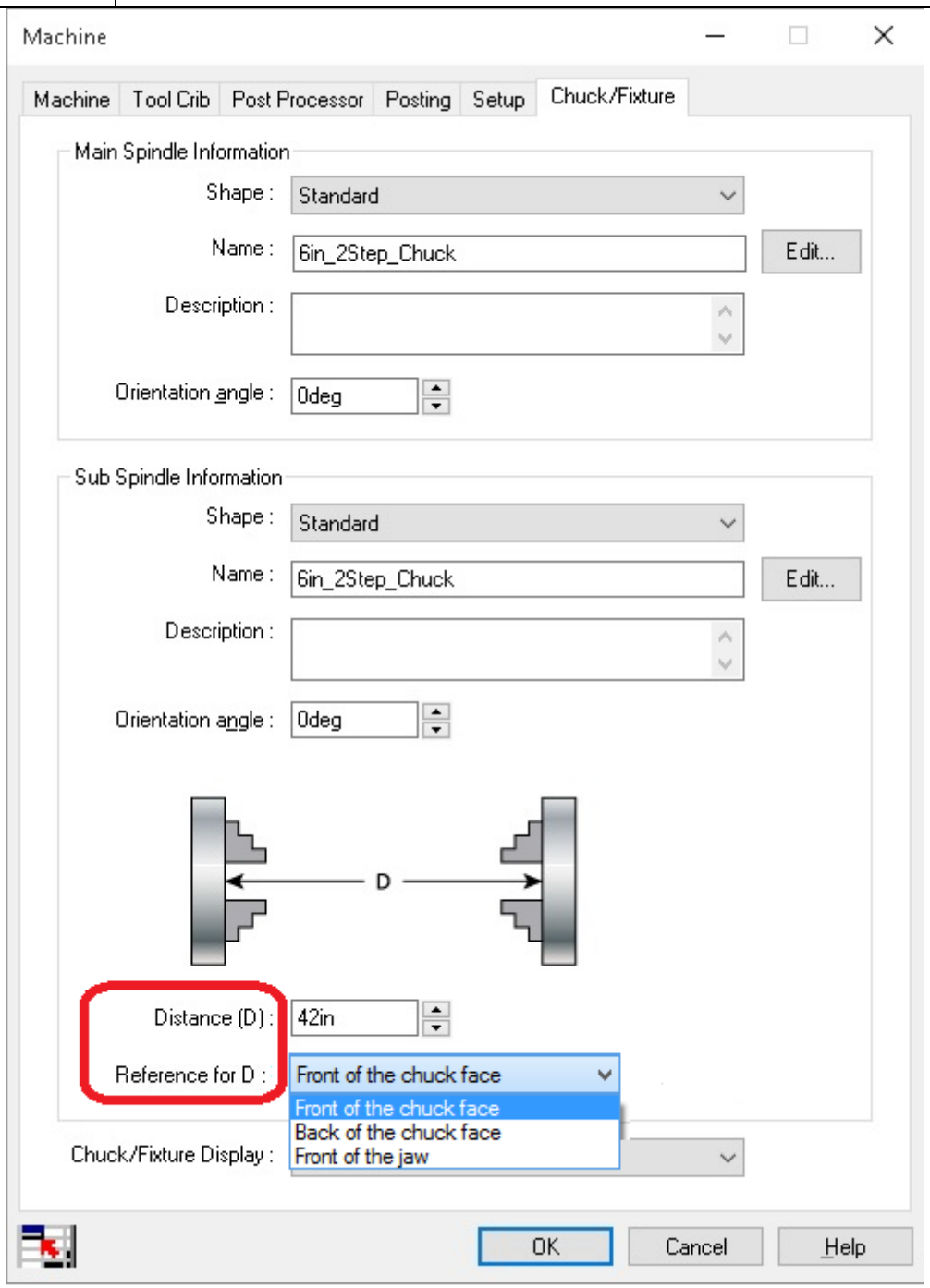
Purpose:	Allow the user to define the start and end location of thread cycle by using the new dropdown options available in Lead In/Out tab of the Operation Parameters dialog box.
Implementation	In previous versions of CAMWorks, these controls were driven from the Windows Registry settings. From CAMWorks 2017 version onwards, new dropdown options are available in the Leadin/Out tab to define the start and end of thread toolpath move. These options allows the user to start or end the thread cycle after or before completing the lead in and out thread toolpath move. **Note:** The ***Start thread cycle at*** option is enabled only when ***Leadin type*** is set to ***Parallel*** or ***Perpendicular***. The ***End thread cycle at*** option is enabled only when ***Leadout type*** is set to ***Parallel***, ***Perpendicular*** or ***Same as leadin***.



Lead In/Out tab of the Operation Parameters dialog box

Improved - Enhanced options for defining the distance between Chucks

Purpose:	Options to define the distance between chucks of the Main Spindle and Sub Spindle
Implementation:	This option is provided in the ***Chuck/Fixture*** tab of the ***Machine Definition*** dialog box. • The parameter ***Distance (D)*** indicates the physical distance between the chuck face of the main and sub spindles. • The ***Reference for D*** dropdown list in this tab allows selection of the reference position for defining the distance between the chucks of Main spindle and Sub spindle. **Note:** The above parameters are active only when the Sub spindle has been defined in the Machine definition tab.



Parameters for defining distance between chuck in Chuck Tab of Machine dialog box

Improved - View/Change the display state options for Chuck/Fixture from Machine node

Purpose:	Options to view/change the display state of the Chuck/Fixture in the graphics area will now also be available in the ***Chuck/Fixture*** tab of the ***Machine*** dialog box.

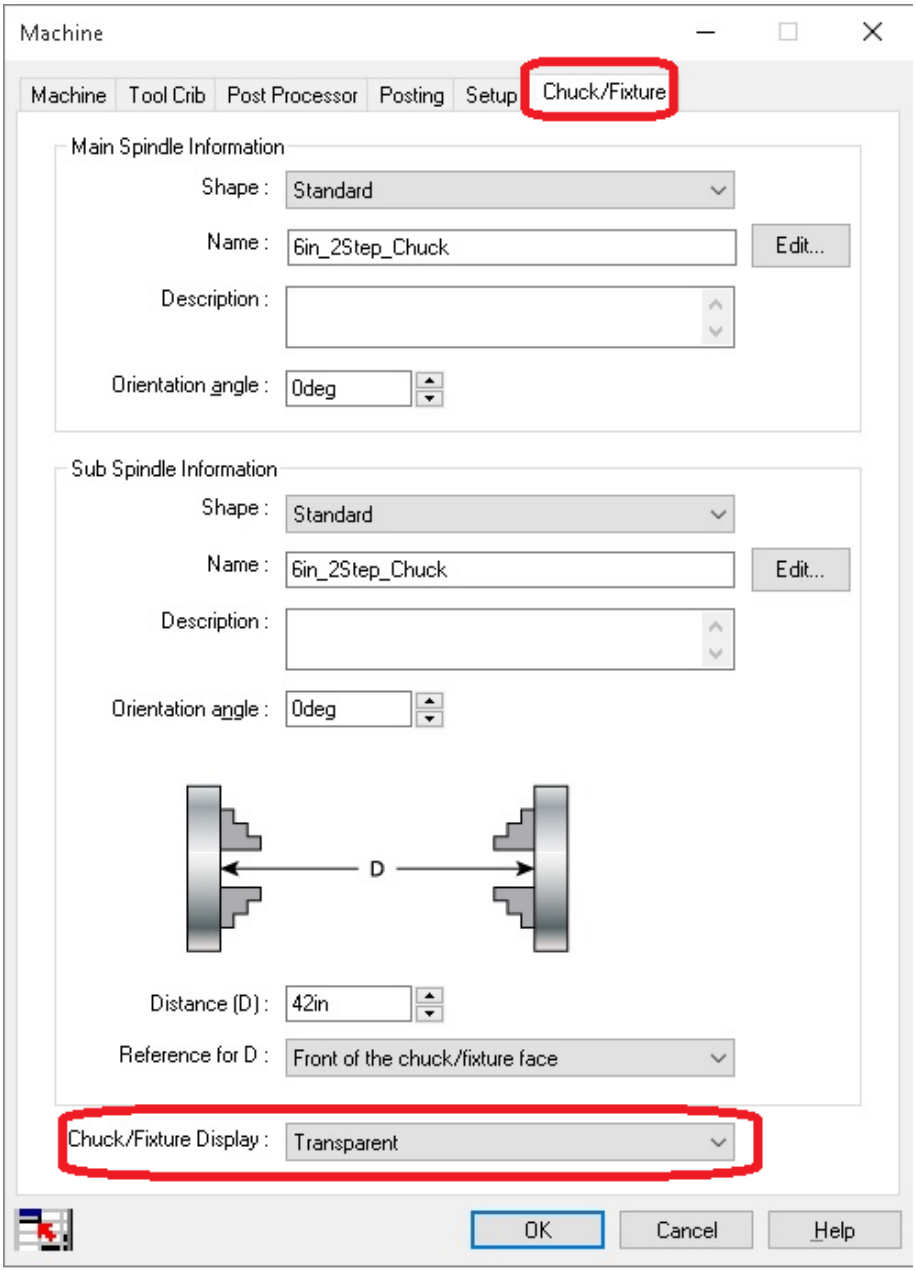
Implementation:

In previous versions on CAMWorks, to change the display state of the chuck/fixture in the graphics area, the following methods were available:

- Right-click on the Machine node in the Feature tree and select the desired display option from the ***Chuck/Fixture Display*** menu.
- Right-click on the Machine node in the Operation tree and select the desired display option from the ***Chuck/Fixture Display*** menu.

From *CAMWorks 2017* version onwards, the following additional method to change the display state of the chuck/fixture is available:

- Open the ***Chuck/Fixture tab of the Machine dialog box*** and select the desired option from the ***Chuck/Fixture Display*** dropdown list.



Chuck/Fixture Display opytions in Chuck/Fixture tab of Machine dialog box

Note:
All the above commands are associative. Settings for display state option of chuck/fixture changed using any one of the above commands will be reflected in the other commands too. For example, if you change the display option for chuck/fixture to *Shaded* using the *Chuck/Fixture Display* context menu of the *Machine* node in CAMWorks Operation tree, then the same setting will be reflected in the *Chuck/Fixture Display* context menu of the *Machine* node in CAMWorks Feature tree as well as ***Chuck/Fixture Display*** dropdown list in ***Chuck/Fixture tab of Machine dialog box***.

APIs and Macros introduced in CAMWorks 2017

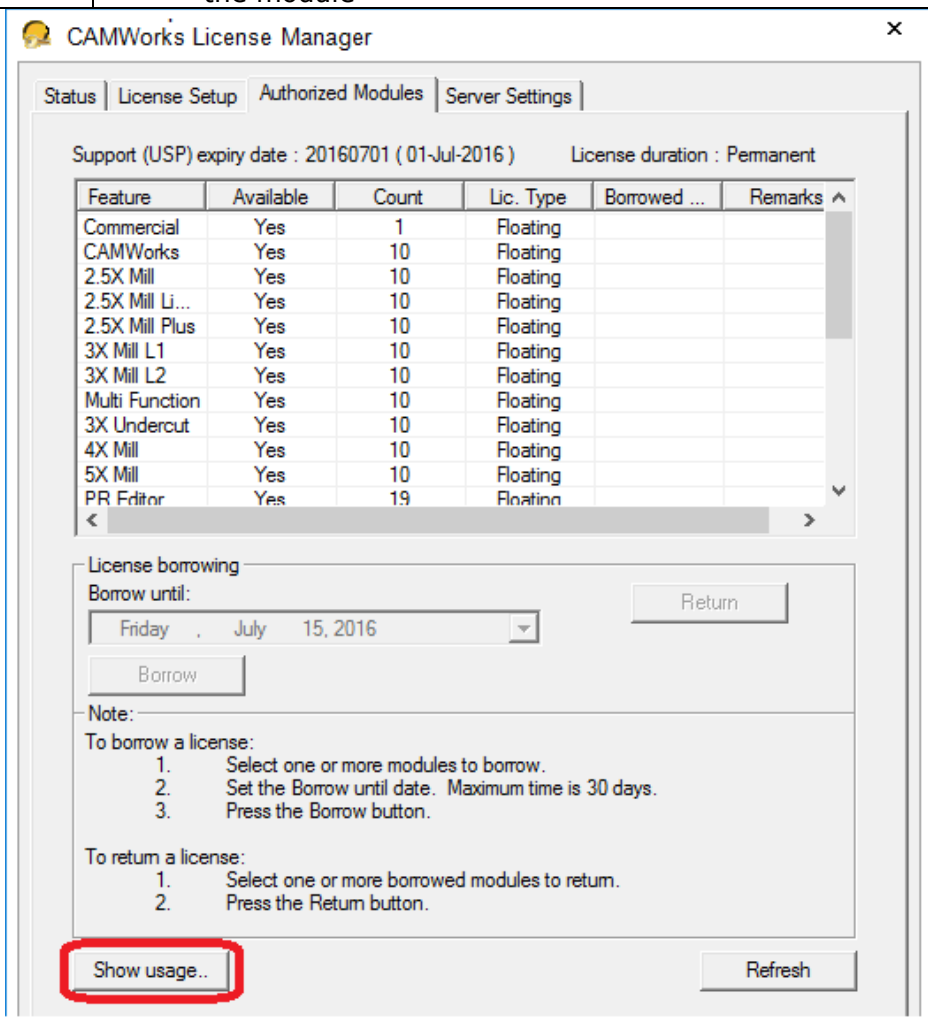
Purpose:	Achieving automation for various functionalities of CAMWorks by use of APIs
Reference document:	To view the list of APIs provided in CAMWorks, along with sample macros provided for APIs introduced in CAMWorks 2017, click on the Windows ***Start*** menu and select ***All Programs>>CAMWorks 2017x64>>Manuals>>What's New in CAMWorks APIs***.

Sr. No.	Interface	Method	Description
1.	ICWMillTool	LenghtOffset	property LenghtOffset
2.	ICWMillTool	DiameterOffset	property DiameterOffset
3.	ICWFeature	GetInnerFeature	method to get inner mill feature
4.	ICWAsmMachine3	IGetActiveWorkpiece	method IGetActiveWorkpiece
5	ICWThreadMillOp2	IGetLeadin	property Get Leadin
6.	ICWThreadMillOp2	IGetLeadout	property Get Leadout
7.	ICWTool	CreateSTLOfCuttingPortion	Creates STL file for Cutting Portion at the specified location
8.	ICWTool	CreateSTLOfNonCuttingPortion	Creates STL file for Non-Cutting Portion at the specified location
9.	ICWMSHoleFeat	NumOfSteps	property get Number of Steps
10.	ICWMSHoleFeat	SetDepth	Method Set Depth for single step Hole feature
11.	ICWPartDoc	PostProcessSelectedOpers	method PostProcess for the Selected Operations"
12.	ICWAdv3xCurveProjectOp	GetZAllowance	Method to Get Z Allowance
13.	ICWFeature	CreateSketchDrivenPattern	Create Sketch Driven Pattern
14.	ICWApp4	GetAppRebuildOptions	method GetAppRebuildOptions
15.	ICWApp4	SetAppRebuildOptions	method SetAppRebuildOptions
16.	ICWOpSetup3,ICWFeature	IGetAllDefaultOperations	Gets all default operations for input operation type
17.	ICWOpSetup3,ICWFeature	GetTdbIdForDefaultOperation	Gets all TechDB ID for input default operation
18.	ICWOpSetup3,ICWFeature	InsertOperation	Method to insert specific operation with default operation ID as an input
19.	ICWOpSetup3,ICWFeature	InsertOperation2	Method to insert specific operation with default operation ID as an input"
20.	ICWOpSetup3	InsertOperationWithTool3	Method to insert operation with specific Tool and with specific default operation
21.	ICWCenterDrillTool	DrillDiameter	property DrillDiameter
22.	ICWTurnToolHolder	GetHolderEdge	To get holder edge from Turn Tool holder
23.	ICWTurnToolHolder	SetHolderEdge	To set holder edge for Turn Tool holder

License

Improved - Tracking usage of licenses for a CAMWorks Floating Network license

Purpose:	Provides the ability to track the active, in use or borrowed licenses on a CAMWorks Floating Network license Configuration
Implementation:	The ***Show Usage*** button is visible in the ***Authorized Modules*** tab of the ***CAMWorks License Manager*** dialog box only when the particular machine has been setup as a client accessing the CAMWorks floating license. It is visible in the ***Server Settings*** tab only when the particular machine has been setup as the ***CAMWorks License Server*** CAMWorks License Manager and the ***CAMWorks License Service*** is running on that machine. When this button is clicked, a resizable Notepad/text window indicating license usage details will be displayed. Following details will be listed for each authorized CAMWorks module: • Total number of licenses issued for the module • Total number of licenses of the module in use • Total number of licenses of the module that have been borrowed • Vendor name • Expiry date • Name of the machines that are currently using the license of the module (along with Port number being used) • Name of the machines that have currently borrowed the license of the module



Show usage button in Authorized Modules Tab