What's New

in

CAMWorks® 2020

An HCL Technologies Product

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

Copyright © 2019-2020 Geometric Americas Inc. All Rights Reserved.

April 2020

Table of Contents

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

What's New in CAMWorks 2020 – SP2

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2020 - SOLIDWORKS 2019 - CAMWorks Solids 2020 - CAMWorks Solids 2019
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] **Note:** CAMWorks 2020 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>>CAMWorks2020x64>>Resolved CPR's.*

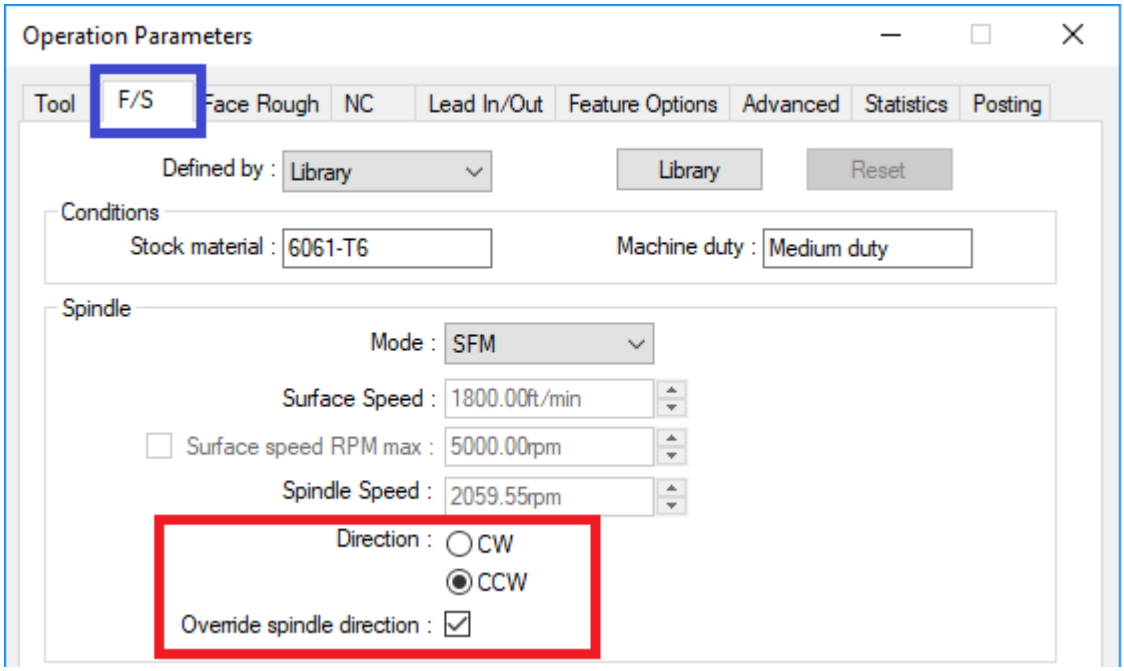
Improved - Behavior of Auto-Spindle Direction Assignment Functionality

Purpose:

To improve the functionality of automatically setting the Spindle Direction (Clockwise or Counterclockwise) of Turn operations

Implementation:

The functionality of automatically assigning of Spindle Direction for Turn operations was introduced in *CAMWorks 2020 SP0* version. This functionality has been tweaked to avoid potential errors especially for legacy parts (viz. parts programmed using any version prior to the *CAMWorks 2020 SP2* version).



'Override spindle direction' checkbox option in F/S tab for a Turn operation

From *CAMWorks 2020 SP2* version onwards, the behavior of this functionality has been altered as follows:

Behavior for Newly Generated Turn Operations:

For all newly generated Turn operations, the spindle direction will be auto-assigned based on the values assigned to parameters that control the spindle direction.
If you wish to override the auto-assigned spindle direction, ensure that the *Override Spindle Direction* checkbox option within the ***Spindle*** group box of the ***F/S*** tab is checked and then change the spindle direction.

Behavior for Legacy Parts (Parts programmed using Version prior to CAMWorks 2020 SP2)

When Turn parts programmed in any version prior to *CAMWorks 2020 SP2* are opened in *CAMWorks 2020 SP2* or later versions, then the user-assigned Spindle Direction settings for the Turn operations of the legacy part will be retained. Retention will be achieved by auto-checking of the ***Override Spindle Direction*** checkbox and assigning the preferred spindle direction (*Clockwise* or *Counterclockwise*).

(The Warning Message that appeared in *CAMWorks 2020 SP0* and *SP1* version when the user-assigned spindle directions of one or more Turn operations were not in sync will no longer be displayed.)

What's New in CAMWorks 2020 – SP1

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2020 - SOLIDWORKS 2019 - CAMWorks Solids 2020 - CAMWorks Solids 2019
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] **Note:** CAMWorks 2020 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>>CAMWorks2020x64>>Resolved CPR's.*

New - APIs introduced in CAMWorks 2020 SP1

Purpose:	Achieving automation for various functionalities of CAMWorks through use of APIs and Macros.
Reference document:	**The 'What's New in APIs' Document** All newly introduced APIs are listed in the ***Whats_New_in_CAMWorks_APIs.pdf*** document. APIs are listed in tabular formats categorized on the basis of the CAMWorks version they were introduced in. This reference document also provides a list of Sample Macros provided with the CAMWorks application along with their purpose and functionalities. **Accessing the 'What's New in APIs' Document** • On ***Windows 7*** machines, this pdf document can be accessed from the Windows ***Start*** menu by selecting ***All Programs>>CAMWorks 2020x64>>Manuals>>What's New in CAMWorks APIs***. • On ***Windows 10*** machines, this pdf document can be accessed from the Windows ***Start*** menu by selecting ***CAMWorks 2020x64>>What's New in CAMWorks APIs***.

What's New in CAMWorks 2020 – SP0

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2020 - SOLIDWORKS 2019 - CAMWorks Solids 2019 - 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 2020 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>>CAMWorks2020x64>>Resolved CPR's.*

Mill

New - Support for Probe Tools and Probing Operations

Purpose:

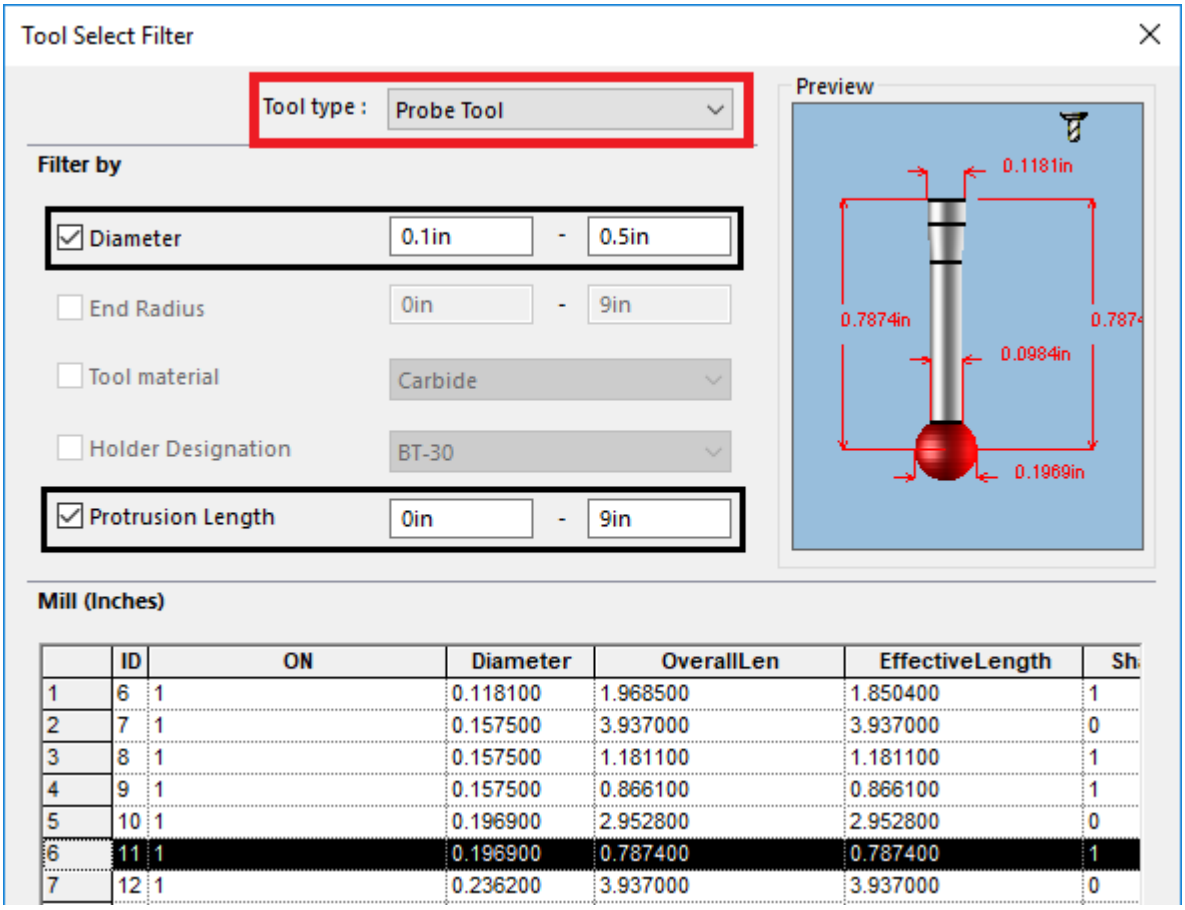
To provide the functionality of Probing Operations using Probe Tools for Milling

Implementation:

Probing is an established best practice for maximizing the efficiency, quality, capability and accuracy of machine tools.
From *CAMWorks 2020* version onwards, Probing operations will be supported in CAMWorks Mill mode.

Probe Tools for Probing Operations

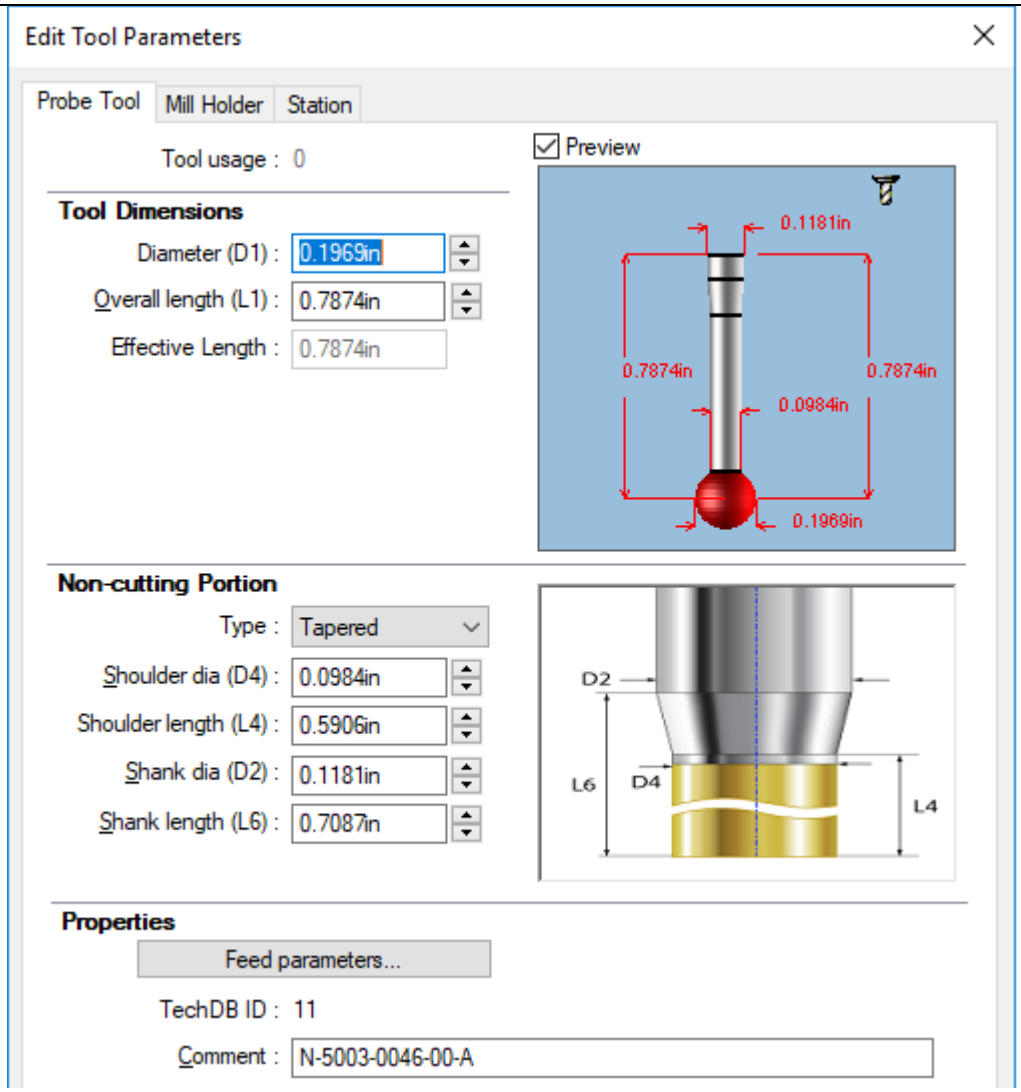
A new type of tool named Probe Tool will be available in the CAMWorks Tool Library. Probe Tools can also be defined and saved in the TechDB. To assign Probe Tools to Probing operations, the desired Probe Tools can be added to the Tool Cribs in the CAMWorks user interface as well as the TechDB.
When adding or replacing tools in the Tool Crib using the ***Tool Select Filter*** dialog box, Probe Tools can be filtered on the basis of their ***Probe Diameter*** and ***Protrusion Length***.



	ID	ON	Diameter	OverallLen	EffectiveLength	Sh
1	6	1	0.118100	1.968500	1.850400	1
2	7	1	0.157500	3.937000	3.937000	0
3	8	1	0.157500	1.181100	1.181100	1
4	9	1	0.157500	0.866100	0.866100	1
5	10	1	0.196900	2.952800	2.952800	0
6	11	1	0.196900	0.787400	0.787400	1
7	12	1	0.236200	3.937000	3.937000	0

'Tool Select Filter' dialog box when a Probe Tool is selected

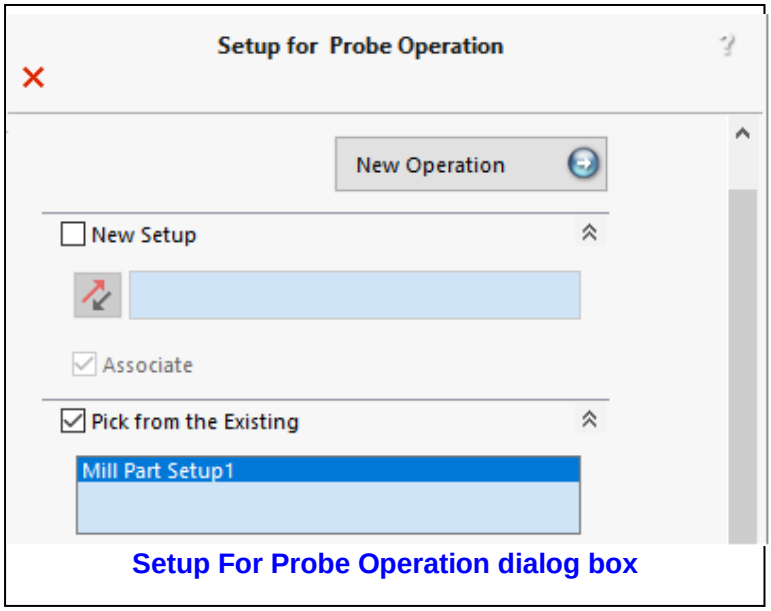
To edit the parameters associated with a Probe Tool, use either the ***Probe Tool*** tab in the ***Edit Tool Parameters*** dialog box or ***Probe Tool*** page under ***Tool*** tab of the ***Operation Parameters*** dialog box for a Probing Operation.



Use the Probe Tool Tab to edit parameters associated with a Probe Tool

Probing Operations

Probing Operations can be interactively inserted by executing the *Insert Probing Operations* command on the CAMWorks Command Manager. Alternatively, this command is also available in the Operation tree in the form of the *Probing Operation* command in the RMB context menu of the *Mill Part Setup* and Operation nodes.
Executing this command displays the *Setup for Probe Operation* dialog box. Use this dialog box to either define a new Setup or pick an existing setup under which the Probing operation is to be listed.

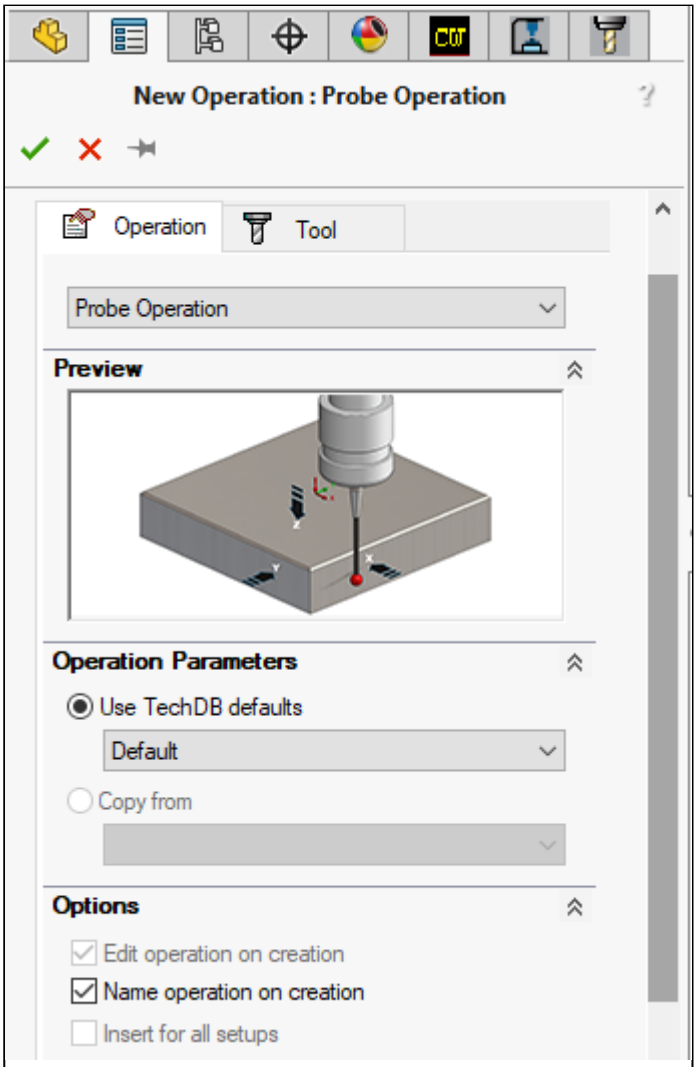


Setup For Probe Operation dialog box

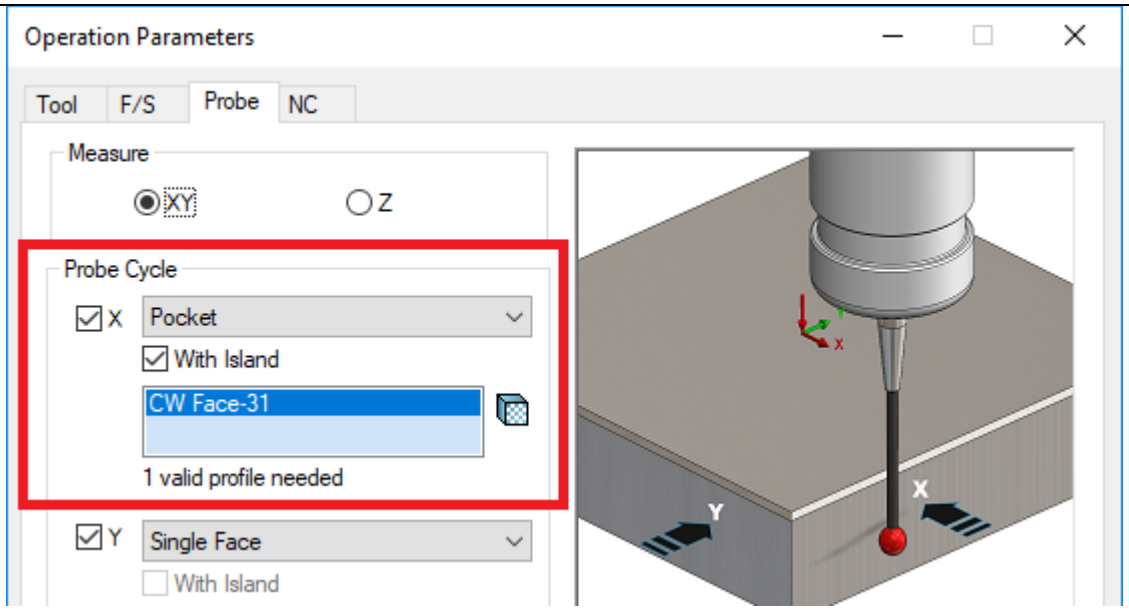
Once the Setup is defined, create a probe operation by selecting the option of "New operation".

To assign the Probe tool to be used for the Probing operation, pick either a Probe tool from the Tool Crib or add a Probe tool from the Tool Library using the options provided in the *Tool* tab.

When you click the OK button  within this interface, the Probe Operation will be inserted in the Operation tree. Depending on the options set for editing operation name and comments, you will get the dialog box to edit the operation name.
Use the *Operation Parameters* dialog box of the Probe Operation to edit the settings for the Probe operation.
In CAMWorks, you can define various types of probe cycles. These include Probing a single face, Web, Pocket, Boss features and Bore features (Holes). Also, the other cycles include probing the boss feature and bore features through 3 points



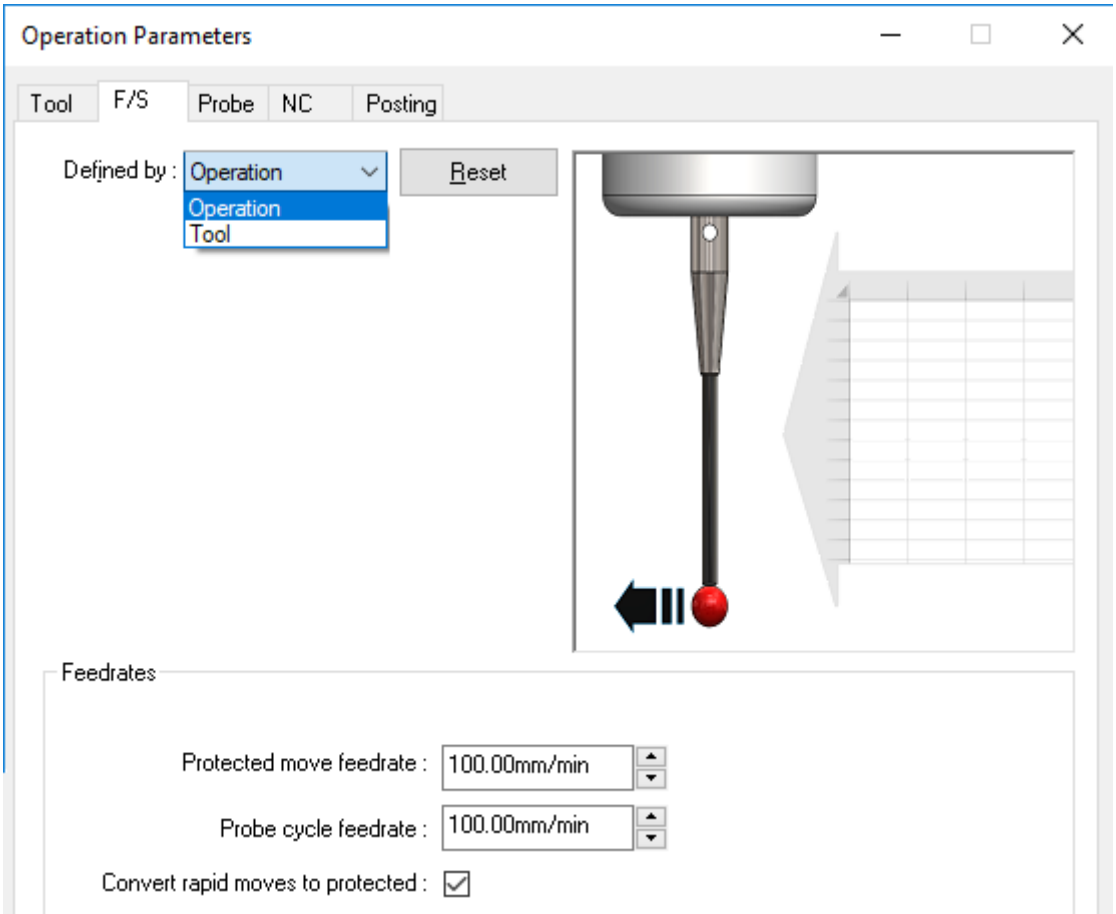
New Operation: Probe Operation Dialog Box



Probe Tab of Operation Parameters dialog box for Probe Operations

Use the ***F/S*** tab to define the federate which the probe will move. This federate can be defined either by the Tool or by Operation.

Two feedrates need to be assigned in the F/S tab for Probe operations - one for Protected moves and the other the actual probe moves.



F/S tab in Operation Parameters dialog box for Probe Operations

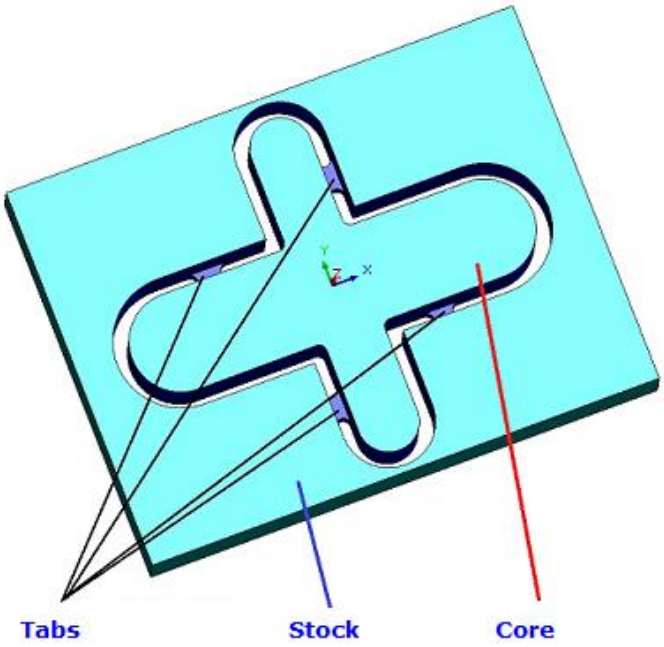
New - Support for Tabs Cutting in Contour Mill Operations

Purpose:

To provides a functionality whereby users can define tabs on the feature profile of a through mill feature machined using a single Contour Mill application.

Implementation:

When machining mill features with an intent to retain the core material with a single contour mill pass, provision must be made so that the core material is retained till the contour milling is done on the entire profile of the feature.
The solution for such an error is by using 'tabs'. Within the purview of CNC machining, a 'tab' is a small piece of the stock material along the profile (feature boundary) of through mill feature which holds the core material to the stock.



Sample Image of a Through Mill feature with Tabs on its Feature Periphery

Tab Settings within the CAMWorks User Interface

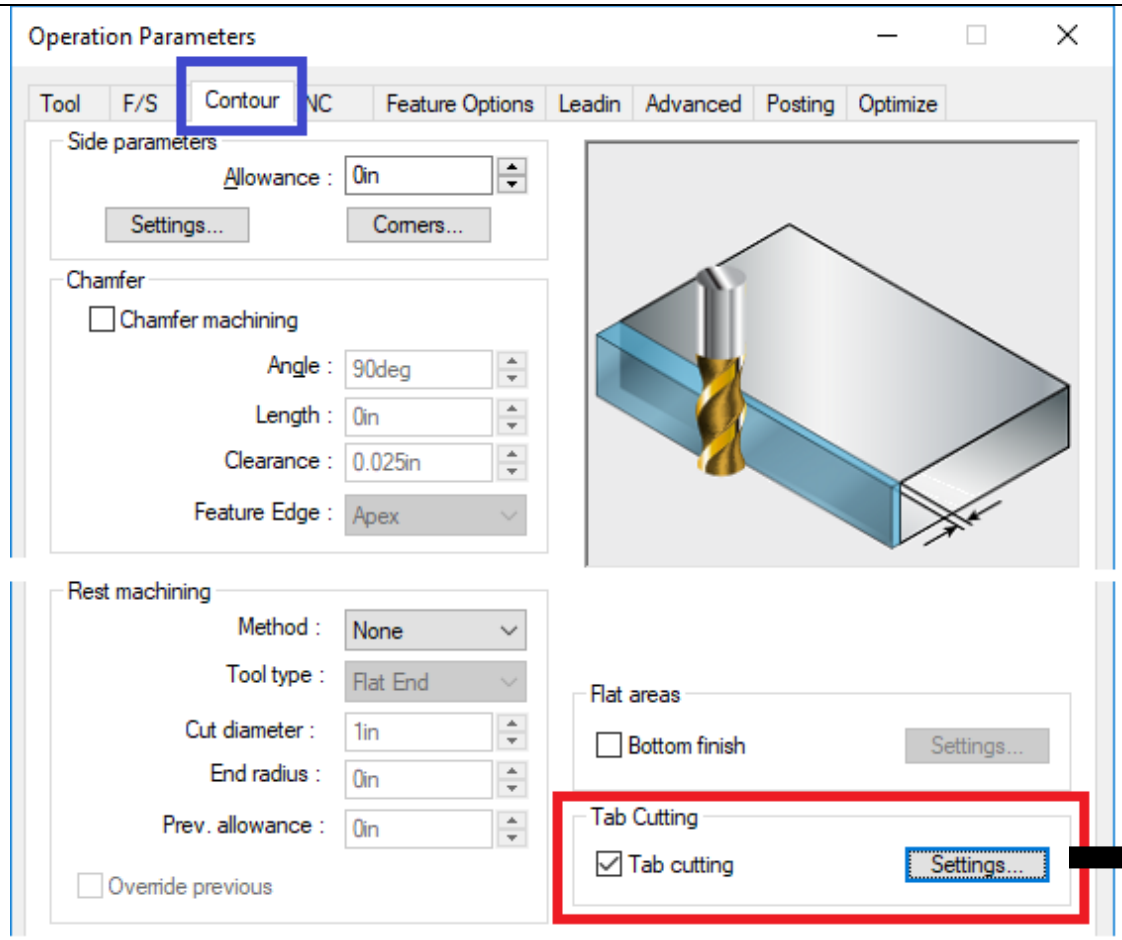
Within the CAMWorks user interface, the option to enable/disable the functionality is provided in the ***Tab Cutting*** group box under ***Contour*** tab of the ***Contour Mill*** operation. To enable the functionality for defining tabs, place a check in the ***Tab Cutting*** checkbox. The ***Settings*** button within this group box will be enabled only when the ***Tab Cutting*** check box is checked. Clicking on this button displays the ***Tab Settings*** dialog box.

Note that the Tabs can be applied only when the option of Bottom Finish is selected.

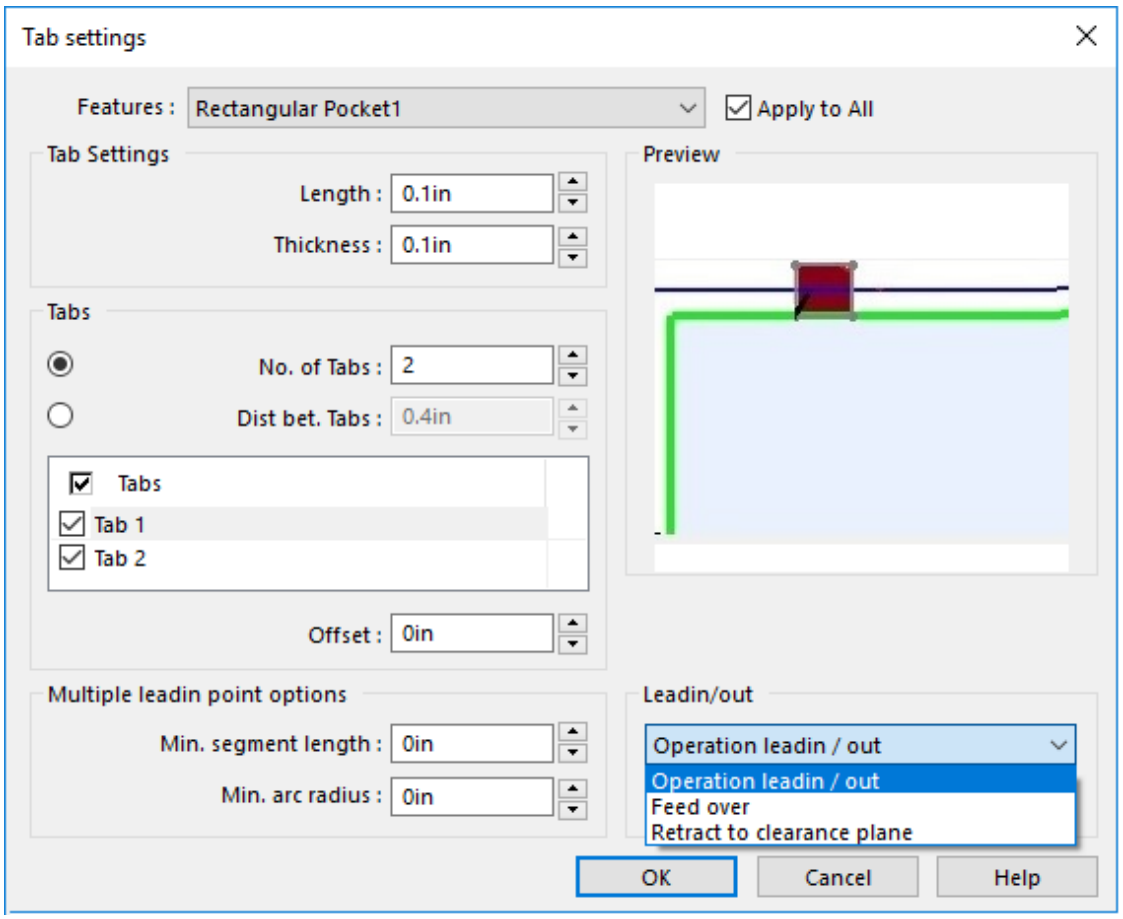
Tab Settings Dialog box

The ***Tab Settings*** dialog box provides the options and controls for defining multiple tabs along the feature profile such that the core is retained with stock. The following settings can be assigned:

- Selecting the feature for which tabs are to be generated (This is applicable only when group features or multiple mill features are machined by the Contour Mill operation)
- Indicating whether the settings in Tab settings dialog box are to be applied to a specific feature being machined by the Contour Mill operation or to all the features being machined by the Contour Mill operation (***Apply to All*** checkbox option)
- The dimensions of the tab (***Length*** and ***Thickness*** parameters)
- Number of tabs to be generated along the feature periphery (***No. of Tabs*** parameter)
- Offsetting the default location of specific individual tabs along the feature periphery (***Offset*** parameter)
- Viewing selected tab in the graphics area (***Tabs*** list box)
- Deselecting specific tabs not be considered when toolpaths are generated. (Checkbox options in the ***Tabs*** list box)
- Impose a filter on the number of tabs that can be generated along the segments/ arcs that comprise the feature periphery (***Minimum Segment Length*** and ***Minimum arc radius*** parameters)
- Defining the exit and entry method for the toolpath so that the tool can retract and enter into the stock material at the tab locations (***Leadin/out*** dropdown list)



'Tab Cutting' group box under Contour Tab of a Contour Mill Operation



Tab Settings Dialog Box

Improved - Advanced Edit Toolpath Dialog Box

Purpose:

To simplify the process of inserting new toolpath records using ***Advanced Edit Toolpath*** dialog box

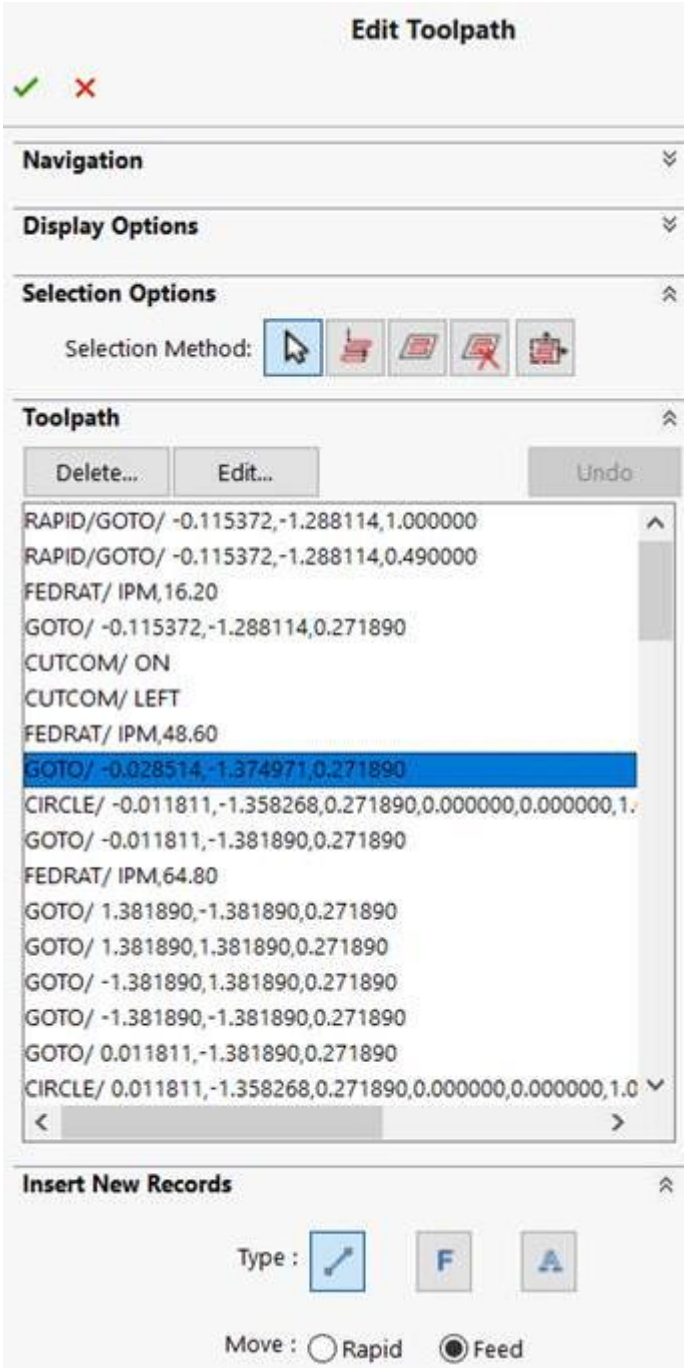
Implementation:

To edit, delete or insert toolpath records associated with a toolpath, the Advanced Edit Toolpath dialog box is provided in the CAMWorks user interface.
This dialog box is displayed when you expand a 2.5 Axis Mill or 3 Axis Mill operation node in the *Operation* tree, right-click on the Mill feature listed under that operation and select the ***Advanced Edit Toolpath*** command from its context menu.

In the previous version of CAMWorks, when the ***Insert*** Button was clicked a separate ***New Toolpath Record*** dialog box would be displayed. Users had to input parametric values associated with the new toolpath record to be inserted in this dialog box. On clicking the OK button within this dialog box, the UI would revert to the ***Advanced Edit Toolpath*** dialog box.

From *CAMWorks 2020* version onwards, the ***New Toolpath Record*** dialog box has been integrated with the ***Advanced Edit Toolpath*** dialog box as the ***Insert New Records*** group box at the bottom of the dialog box.
The layout within the ***Insert New Records*** group box has been simplified into three tabs to correspond to the type of toolpath record to be inserted. The parameters displayed within the ***Insert New Records*** group box depends on the tab that is currently active.
Following are the tabs:

-  : Click on this tab to insert a Linear Rapid move or Feed move.
-  : Click on this this tab to insert a new federate record.
-  : Click on this tab to insert an alphanumeric text record.



Advanced Edit Toolpath Dialog Box

New - Tapered Multi-Point Thread Mill tools now supported

Purpose:

To support machining of Thread Mill Toolpaths using Tapered Multi-point Thread Mill Tools

Implementation:

In previous versions of *CAMWorks*, tapered multi-point thread mill tools were not supported. From ***CAMWorks 2020*** version onwards, tapered multi-point thread mill tools will be supported. Hence, thread mill toolpaths for circular boss, circular pocket and hole features can now be machined using tapered multi-point thread mill tools.

For a ***Thread Mill*** operation machined with a ***Tapered Multi-Point Thread Mill*** tool, the following parameters in its ***Operation Parameters*** dialog box will be affected:

i. In the ***Multi-Point Thread Mill Tool*** page under ***Tool*** tab:
   a. The ***Thread Angle*** parameter will be enabled when ***Type*** in the ***Non-Cutting Portion*** group box is set to ***Tapered***. This parameter defines the angle of the multi-point tapered thread mill as defined from the vertical.
   b. The ***Shoulder Diameter*** will be equal to the inner diameter of the largest thread on the tool.

ii. In the ***Thread Parameters*** group box under the ***Thread Parameters*** tab:
   a. The ***Taper angle*** parameter will be disabled and will be assigned the value defined for the ***Thread Angle*** parameter in the ***Multi-Point Thread Mill Tool*** page under ***Tool*** tab.
   b. The ***Angular Resolution*** parameter will be enabled when the value assigned to the ***Thread Angle*** parameter is greater than zero. This parameter defines the angular resolution or smoothness of the point-to-point toolpath by specifying the maximum angular rotation of each move, as seen from the XY plane. If the angle assigned to this parameter is X degrees, then the number of point-to-point linear moves in the thread mill toolpath will be (360/X).

New - Option to output G-code for Mill Operations in Subroutine format

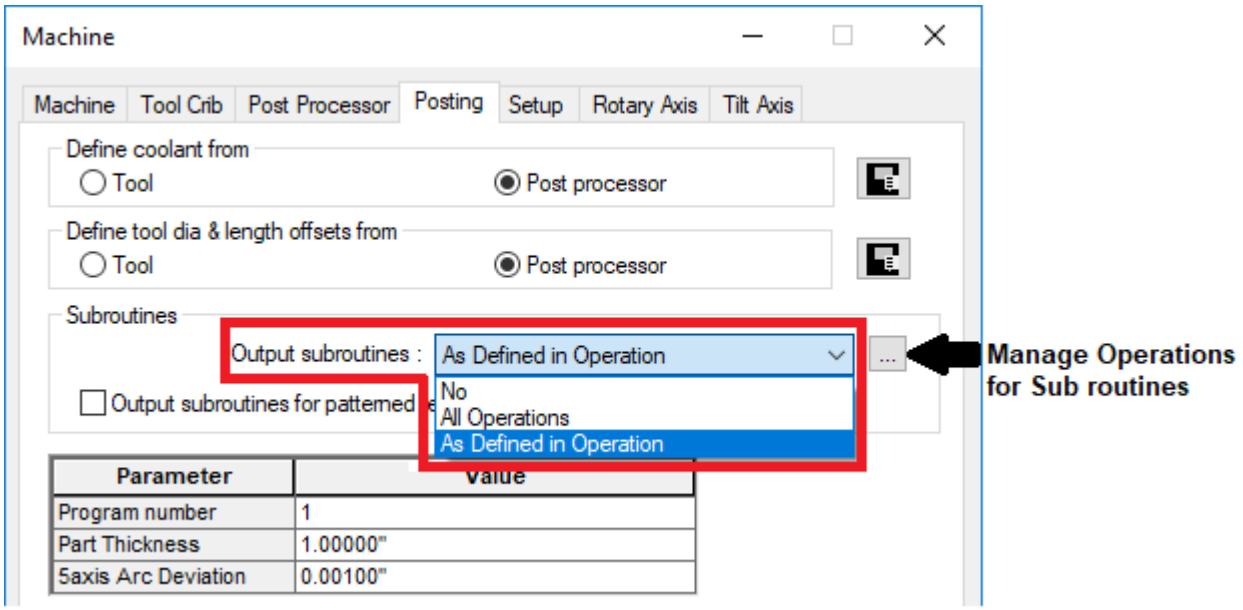
Purpose:

To provide an option whereby the posted G-code for all or specific Mill operations selected by the user can be output in subroutine format

Implementation:

In previous version of CAMWorks, the option to output the G-code in subroutine format was available only for the operations generated for pattern features (in Part and Assembly mode) and part instances (in Assembly mode). (This was provided in the form of the ***Output Subroutine for pattern features*** checkbox option in the *Posting* tab of the ***Machine*** dialog box.) For all other operations, the G-code was output in long code format.

From *CAMWorks 2020* version onwards, the functionality to output G-code in subroutine format has been extended to all operations. This functionality is available in the CAMWorks user interface in the form of the ***Output Subroutine*** dropdown list within the *Posting* tab of the ***Machine*** dialog box.

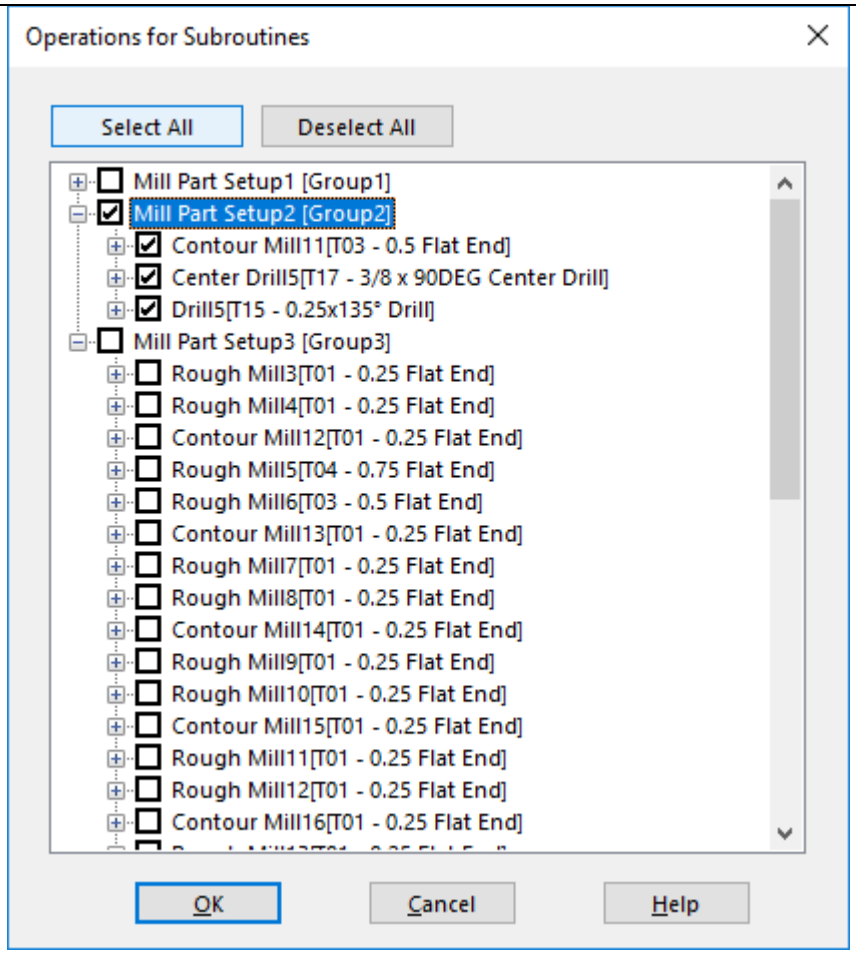


'Output subroutines' dropdown list parameter in Posting tab of Machine Dialog Box

The following options are available within this dropdown list for determining whether G-code for the generated operations are to be output in subroutine format or not.

- **No**: When this option is selected, the posted G-code for all operations will be in long code format only.
- **All Operations:** When this option is selected, the posted G-code for all operations will be in subroutine format. (This will also include pattern features.)
- **As Defined in Operation**: Use this option to specify the operations for which G-code is to be output in subroutine format. When this option is selected, the ***Manage Operations for Subroutines*** button will be enabled.

  Clicking on this button displays the ***Operations for Subroutine***s dialog box. This dialog box lists all the generated operations. Use this dialog box to select the specific operations for which G-code is to be output in subroutine format. To select an operation within this dialog box, expand the ***Mill Part setup*** under which it is listed and place a check in the checkbox adjacent to the operation.



'Operations for Subroutines' dialog box

Improved - Display of Automatically Defined Contain Areas for 3 Axis Mill Operations

Purpose:

To enable display of Automatically Defined Contain Areas for 3 Axis Mill operations in the graphics area

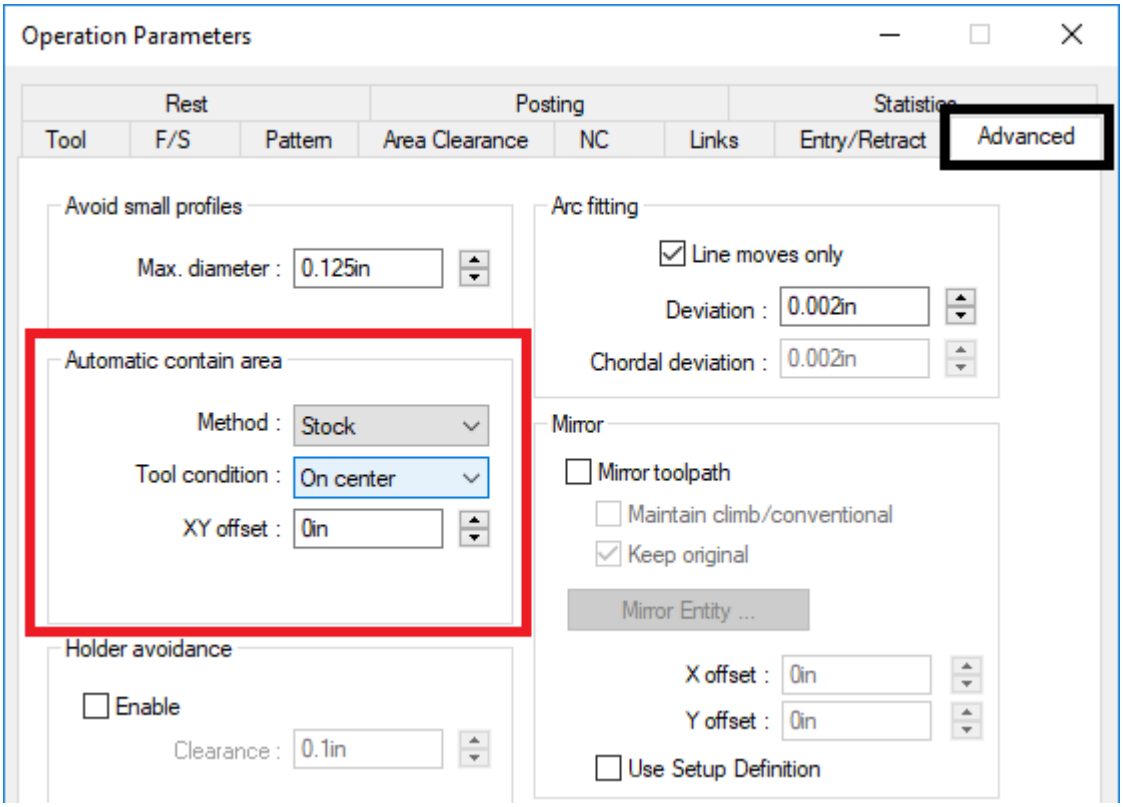
Implementation:

For 3 Axis Mill Operations, the contain area can be automatically defined using the parameters in the ***Automatic Contain Area*** group box under the ***Advanced*** tab in the Operation Parameters dialog box for 3 Axis Mill operations. This functionality for automatically defining contain areas is available for the following operations:

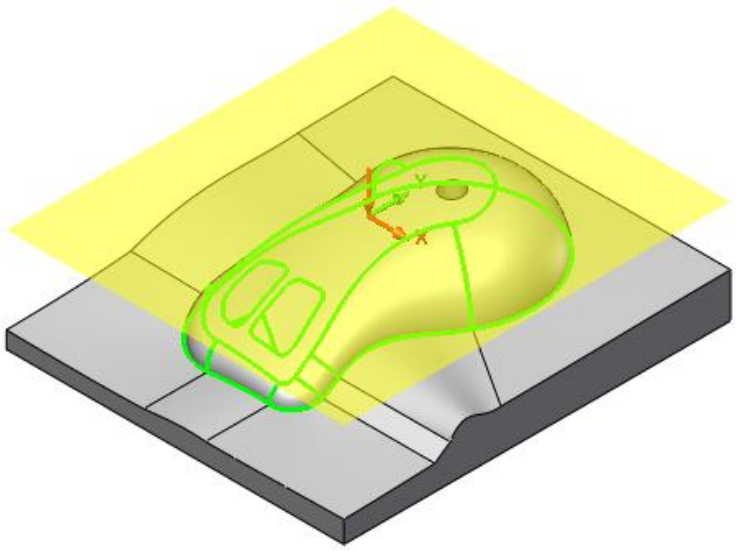
- Area Clearance Operations with *Adaptive* pattern
- Constant Stepover operations
- Pattern Project operations
- Pencil Mill operations

From ***CAMWorks 2020*** version onwards, if automatically defined contain areas have been defined for any of the 3 Axis Mill Operations listed above, then they can be viewed in the graphics area when the following conditions are fulfilled:

- The *3 Axis toolpath generation method* in the *Update* tab of the *CAMWorks Options* dialog box is set to *Advanced method*.
- The *Operation Parameters* dialog box for the 3 Axis Mill operation is open and the cursor focus is on any one of the parameters within the *Automatic Contain Area* group box under its *Advanced* tab.



Automatic Contain Area group box under Advanced Tab in Operation Parameters dialog box



Example of a Automatically Defined Contain Area displayed in the Graphics Area when the cursor focus is on a parameter within the Automatic Contain Area group box in Advanced tab

New - Support for Non-Center Cutting Tools in Mill

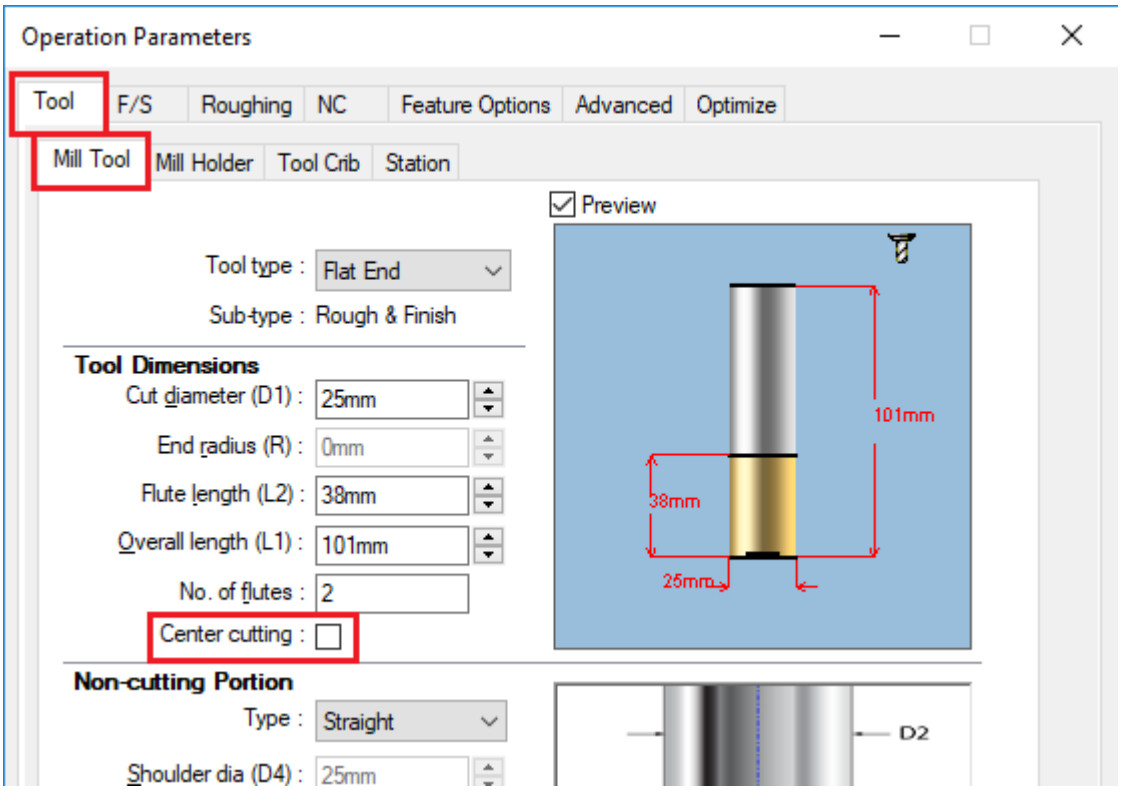
Purpose:

Allows users to define a selected Milling tool as a either a center cutting or non-center cutting tool

Implementation:

The geometry of some milling tools (especially Flat End Mills and Hog Nose Mills) can occasionally have a tool geometry that prevents its center portion from cutting/ machining any material.
In previous versions of CAMWorks, there was no option available to define a mill tool as a non-center cutting tool.

From ***CAMWorks 2020*** version onwards, the option to define a mill tool as a center-cutting or non-center cutting tool has been introduced. This is provided in the form of the ***Center Cutting*** checkbox option in the ***Tool*** page under ***Tool*** tab of ***Operation Parameters*** dialog box.



'Center Cutting' checkbox option in Mill Tool page under Tool tab of Operation Paramters Dialog Box

- When the ***Center Cutting*** checkbox option in the ***Mill Tool*** page is checked, the tool will be treated as Center Cutting tool. (Default Setting)
- When the ***Center Cutting*** checkbox is unchecked:
  i. The tool will be treated as Non-center Cutting tool.
  ii. The tool display in the graphics area will distinctly display the non-cutting portion.

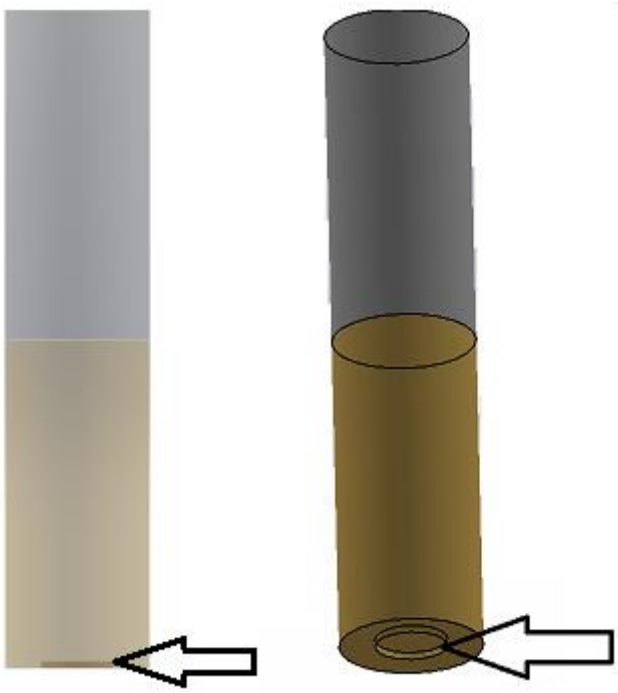
Note:

For Non-center cutting tools (i.e. tools for which ***Center Cutting*** checkbox is unchecked), it's recommended that the ***Entry Method*** option of Plunge not be used for any of the mill operation for which such a tool is assigned. If this ***Entry Method*** is selected for a non-center cutting tool, CAMWorks will display a warning message asking you whether you wish to continue (Yes/No). Clicking ***Yes*** will allow you to continue while clicking ***No*** will bring the focus of the cursor to the ***Entry Method*** dropdown list in the ***Feature Options*** tab.

Center Cutting checkbox option is supported for the following Mill Tool Types:

- Flat End Mill
- Taper Flat End Mill

- Hog Nose
- Taper Hog Nose
- Face Mill
- Keyway
- Dovetail
- Corner Round
- User Defined



Illustrative image of a Mill Tool with its Non-center Cutting portion clearly demarcated (when Center Cutting chckbox option is unchecked)

How Center Cutting Tool Option can affect Toolpath Simulation

If the ***Center Cutting*** option for a mill tool is unchecked and Toolpath Simulation is performed for a part/assembly, then a warning message will be displayed if the non-cutting portion of the tool collides with the stock material. This message will however be shown only if the ***Pause on Collision*** option for simulation is enabled.

New - Curve Features for Chamfering

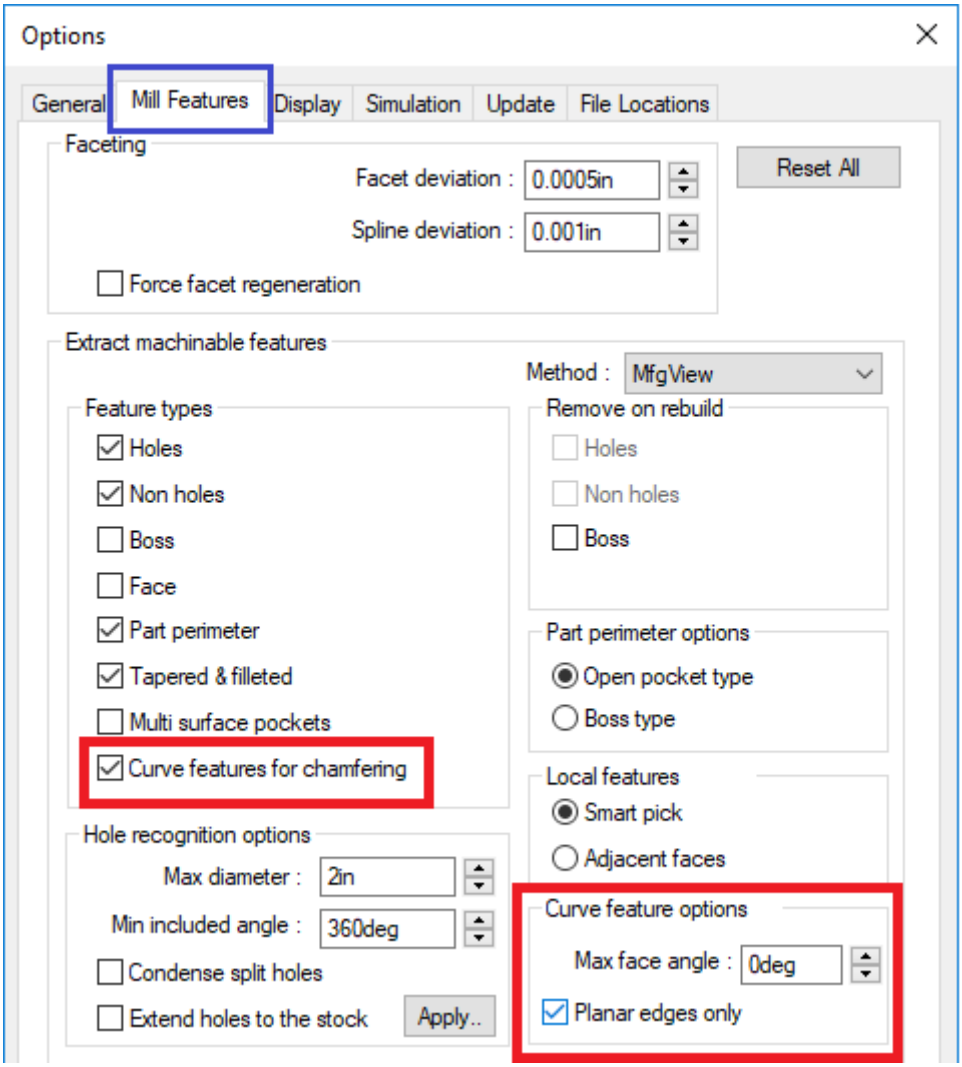
Purpose:

To provide option to recognize *Curve Features for Chamfering*

Implementation:

From *CAMWorks 2020* version onwards, the option to generate ***Curve features for Chamfering*** using *Automatic Feature Recognition (AFR)* is available.
To exercise the option of recognizing *Curve features for Chamfering*, select the option of *Curve Feature for Chamfering* box in the *Mill Features* tab of the *CAMWorks Options* dialog box. Also note that you can provide additional conditions when defining the Curve feature for chamfering using options in the ***Curve Features Options*** group box.

- Use the ***Max Face Angle*** parameter to select the edges for Curve feature. Any edges formed by faces which have an angle exceeding the defined value will not be considered for detection.
- Select the ***Planar edges only*** checkbox option if only edges that lie in the XY plane with respect the Mill Part Setup's origin are to be considered when recognizing the *Curve Feature for Chamfering* using *AFR*.



Option for 'Curve features for chamfering' in the Mill Features Tab of CAMWorks Options Dialog Box

Once the *Extract Machinable Features* command is executed, the convex edges of the part model, as visible from the Mill Setup direction, will be recognized as a *Curve Feature for Chamfering*. Both open-ended and closed ***Curves features for Chamfering*** can be recognized in this manner.

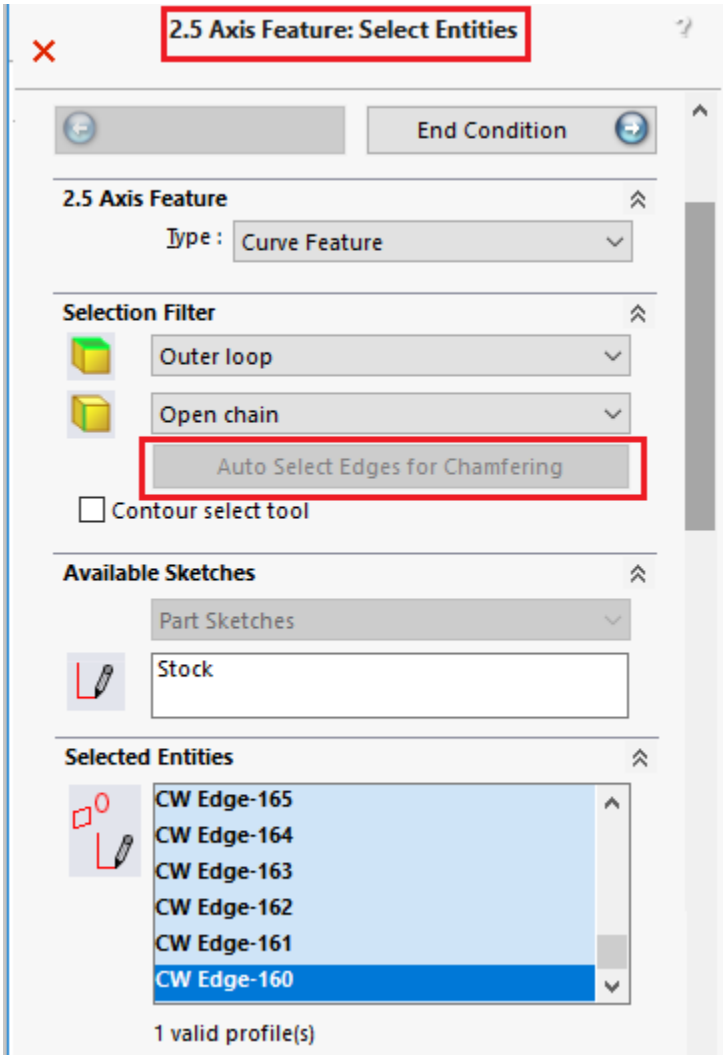
Interactively Inserting Curve Features for Chamfering

If required, Curve Features for chamfering can be interactively inserted using the ***2.5 Axis Feature Wizard***.
When the *Feature Type* selected in the ***Select Entities*** dialog box of this Wizard is a ***Curve Feature***, the ***Auto Select Edges for Chamfering*** button will be displayed within the dialog box.

Executing this button command will auto-detect the convex edges of the part model (as visible from the Mill Setup direction) for chamfering/ edge breaking. The auto-detected edges will be listed in the ***Selected Entities*** list box.

Note:

- The *Auto Select Edges for Chamfering* button will become disabled when the ***Selected Entities*** list box contains any selections.
- The checkbox option ***Check for Tapers and Fillets*** will not be available when the Feature Type selected in the *Select Entities* dialog box is a curve feature.



'Select Entities' dialog box in 2.5 Axis Feature Wizard

New - Display of Clearance Planes for Multiaxis operations in Graphics area

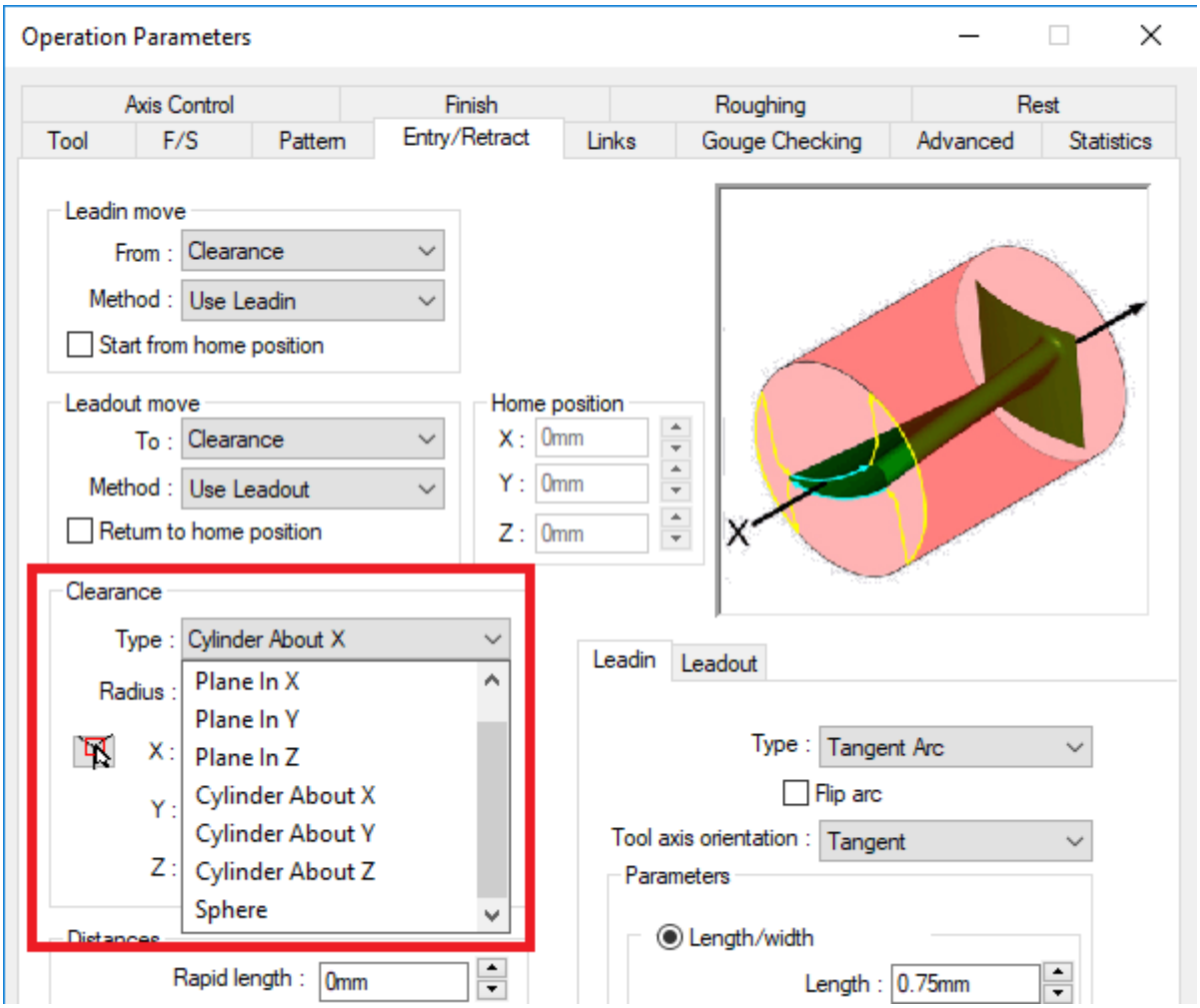
Purpose:

To provide a visual feedback of the clearance plane generated for Multiaxis (5 Axis) operations (includes Multiaxis Mill, Multiaxis Drill and Swarf Milling operations) by displaying the clearance plane in the graphics area

Implementation:

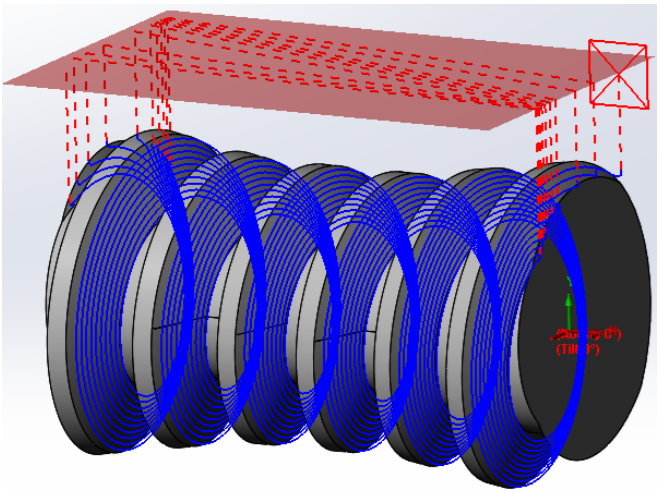
For Multiaxis operations, the *Clearance* plane is defined using the parameters in the ***Clearance*** group box of the ***Entry/Retract*** tab in the ***Operation Parameters*** dialog box for those operations. The following options are available in the ***Type*** dropdown list within this group box to define the type of Clearance plane to be used:

- Plane in X
- Plane in Y
- Plane in Z
- Cylinder about X
- Cylinder about Y
- Cylinder about Z
- Sphere

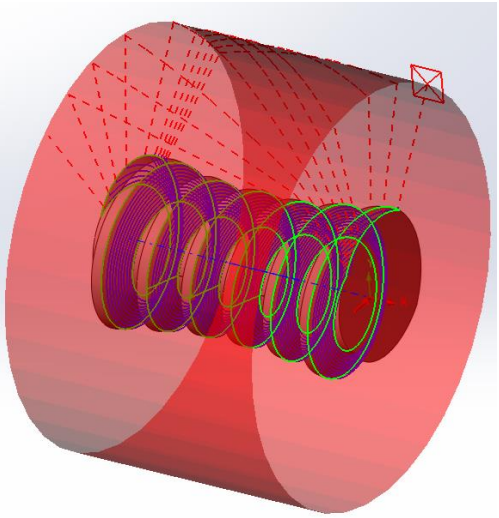


Setting the Clearance Type in Entry/Retract Tab for a Multiaxis Operation

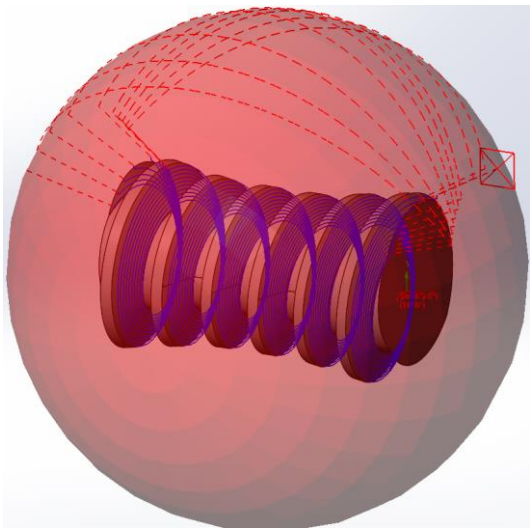
In previous versions of *CAMWorks*, there was no functionality to view the clearance plane of a Multiaxis operation in the graphics area of ***SOLIDWORKS/ CAMWorks Solids***.
From ***CAMWorks 2020*** version onwards, whenever the cursor focus is on the ***Clearance Type*** dropdown list in the ***Entry/Retract*** tab for a Multiaxis operation, and any one of the options listed above is selected in ***Clearance Type*** dropdown list, the corresponding clearance limit will be displayed in the graphics area. This functionality ensures that you get a visual feedback of the clearance plane within the graphics area.
The displayed clearance plane will disappear from the graphics area when you shift the cursor focus to another parameter. The color of the clearance limit can be set by assigning the desired color to the ***Rapid Toolpath*** item in the ***Color Settings*** list box under ***Display*** tab of the ***CAMWorks Options*** dialog box.



Clearance Type set to 'Plane in Y'



Clearance Type set to 'Cylinder about X'



Clearance Type set to 'Sphere'

Turn/Mill-Turn

New - Associating Spindle Direction with the Hand of Cut of the Tool

Purpose:

To provide a mechanism wherein the Spindle Direction (Clockwise or Counterclockwise) is automatically assigned to a Turn operation based on certain tool parameters and thereby eliminates the possibility of incorrect assignment of spindle direction by the user

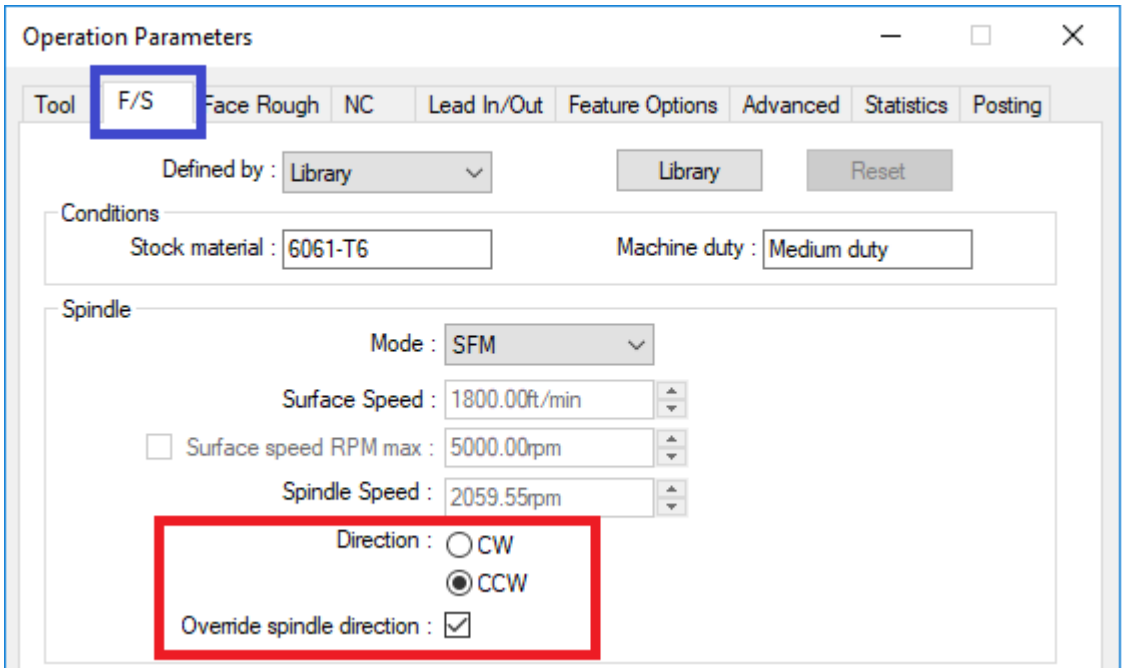
Implementation:

In previous versions of CAMWorks, the spindle direction could be assigned by users using the options for the ***Direction*** parameter in the ***Spindle*** group box under the ***F/S*** tab.

From *CAMWorks 2020* version onwards, the spindle direction parameter will be disabled for all Turn operations under default settings. This direction (***Clockwise*** or ***Counterclockwise***) will be automatically assigned by *CAMWorks* based on the settings assigned to following Turn tool parameters that control the spindle direction:

- The selected ***turret*** (whether the Front turret or Rear turret is selected)
- The selected spindle (whether the Main Spindle or Sub Spindle is selected)
- The ***Hand of Cut***
- The turn feature being machined (OD, ID, Face, Cut Off, etc.)
- The ***Holder Orientation***
- Whether the ***Mirror about centerline*** checkbox option is checked or not

If you wish to override the default spindle direction assigned by CAMWorks for a specific Turn operation, then place a check in the ***Override spindle direction*** checkbox in the ***F/S*** tab for that operation. Checking this option enables the ***Direction*** parameter and changes the previously assigned direction.



'Ovveride spindle direction' checkbox option in F/S tab for a Turn operation

Spindle Direction for Turn/Mill-Turn parts programmed using previous versions of CAMWorks

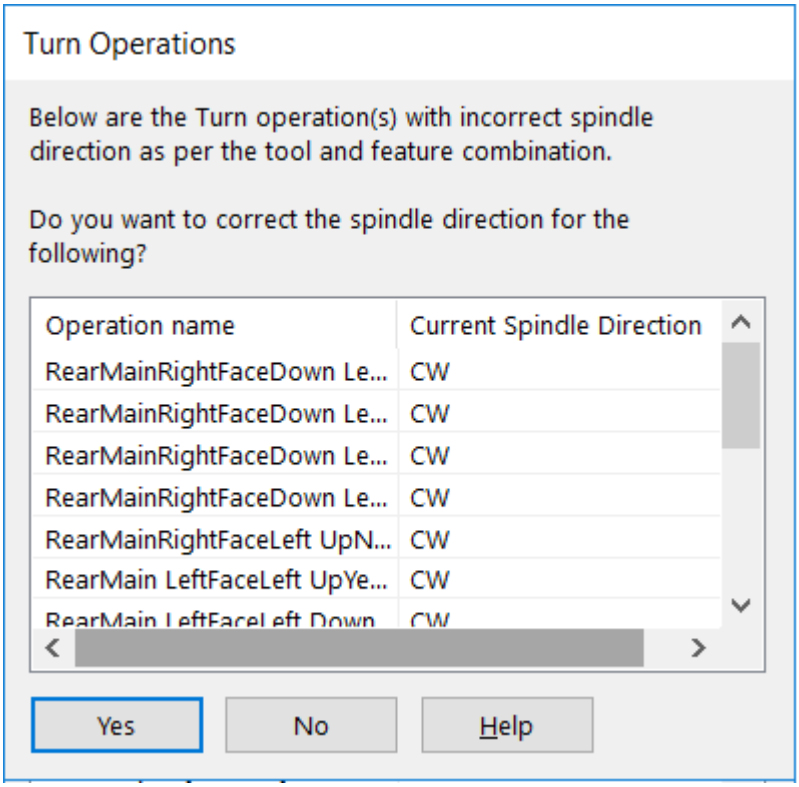
If you open a Turn or Mill-Turn part programmed using such an older version of *CAMWorks* in the *CAMWorks 2020* or any future versions, the following will happen:

i. CAMWorks will verify if the spindle directions assigned to the Turn operations are in sync with the parameters that control the spindle direction.

ii. If all the spindle direction of all the Turn operations are in sync, then no changes will occur to the assigned spindle directions.

iii. If the spindle direction of one or more Turn operations are not in sync, then CAMWorks will display a warning message stating that the spindle directions of specific Turn operations are not in sync with the parameters that control the spindle direction. The names of all the Turn operations and their corresponding corrected spindle directions will be listed within this message box.

- If you click ***Yes*** within this message box, then the message box will close and CAMWorks will change the spindle direction of all the Turn operations listed within the message box to ensure they are in sync with the parameters that control the spindle direction. These changes made automatically to the spindle direction by CAMWorks will be saved when you save the part.
- If you click ***No*** within this message box, the spindle directions for none of the Turn operations listed within the message box will be changed. The Override Spindle Direction checkbox option in the F/S tab will be checked for each of the listed Turn operations. These changes will be saved when you save the part.



Sample CAMWorks Message indicating that the Spindle Direction of certain Turn operations are not in sync with tool and feature combination

Technology Database

New - Options to Save and Restore customized Tabular Grid Display in TechDB UI

Purpose:

To provide options to edit the tabular layouts within the TechDB and save the customized layouts so that users can consume the displayed data as per their desired custom settings

Implementation:

Within the TechDB user interfaces, parameters pertaining to various entities (like features, operations, tools, machines, etc.) is displayed in a tabular grid format. These grids can be customized to suit your display requirements. These settings include:

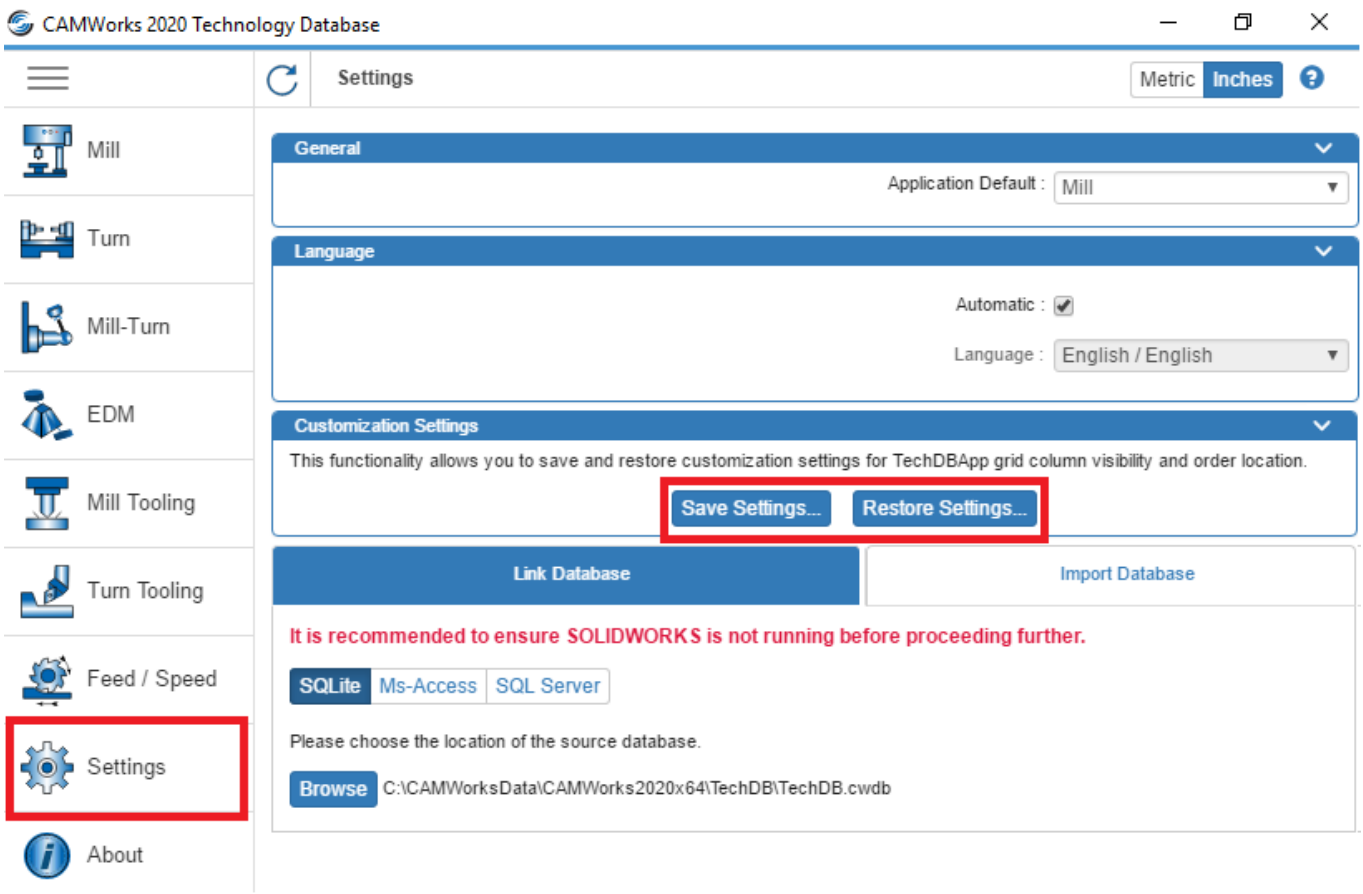
- Adjusting the width of the columns
- Rearranging the order of the columns
- Displaying only specific columns

The custom grid display settings are loaded whenever you launch the TechDB. They are retained even when data is imported from another TechDB.

From *CAMWorks 2020* version onwards, CAMWorks provides a functionality whereby:

1. The custom settings for the grid display can be saved as an external file (*.cwjs)
2. The grid layout of any other TechDB App can be customized by restoring/applying a previously saved external *.cwjs file

This functionality is provided in the form of the ***Save Settings*** and ***Restore Settings*** buttons in the ***TechDB Settings*** user interface.



The 'Save Settings' and 'Restore Settings' button commands in the TechDB Settings User Interface

Additive Manufacturing

New - 3D Printing of Assemblies using Additive Manufacturing Module of CAMWorks

Purpose:

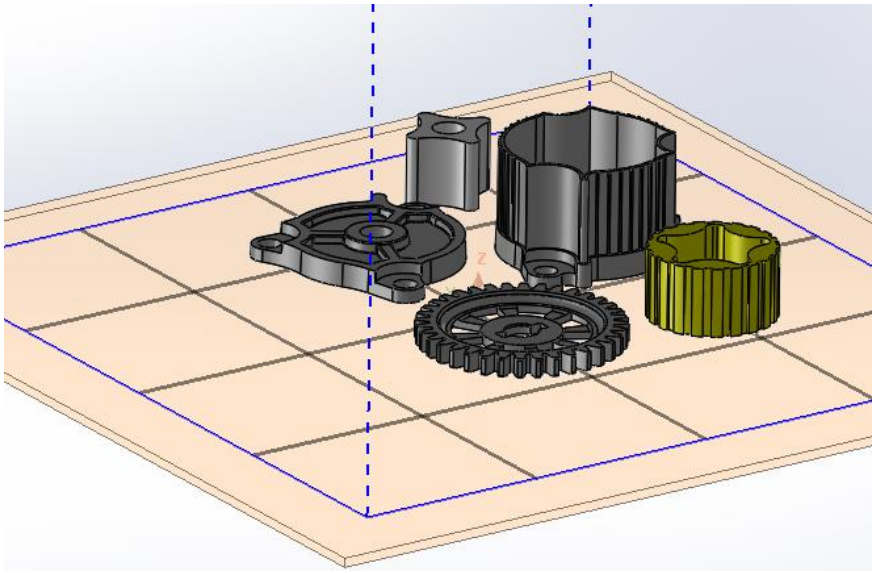
To extend the functionality of 3D printing using the ***CAMWorks Additive Manufacturing*** module to Assemblies

Implementation:

In previous versions of the ***CAMWorks Additive Manufacturing*** module, part files could be 3D printed by generated Build Tickets using the ***Create AM Job Wizard***.
From *CAMWorks 2020* version onwards, this functionality has been extended to Assemblies too.

Pre-requisites for executing an Additive Manufacturing Job for an Assembly

- All those parts of the assembly that are to be 3D printed must fit within the machine volume
- All those parts of the assembly that are to be 3D printed must be aligned with the build plate.



Example of an Assembly whose component parts have been aligned with the Build Plate for Additive Manufacturing

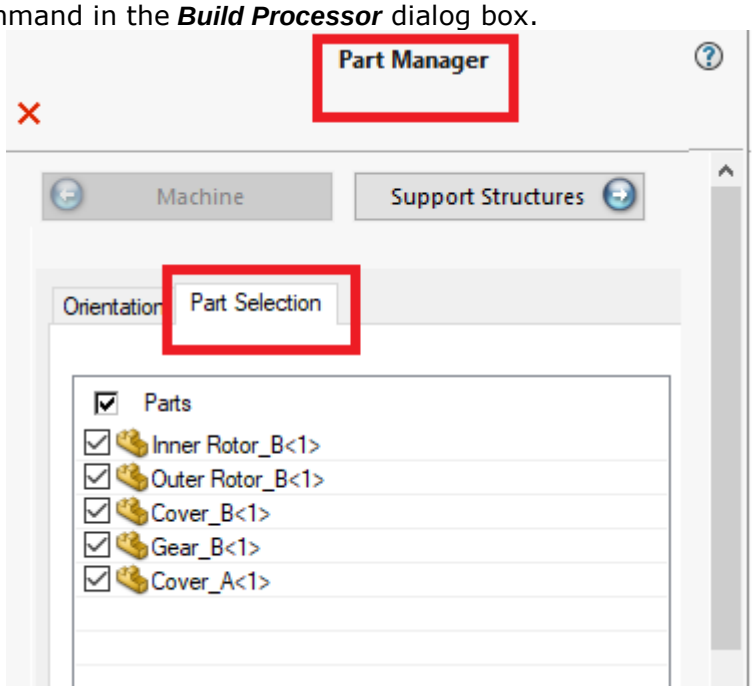
Enhancements in the CAMWorks Additive Manufacturing User Interface for supporting 3D Printing of Assemblies

The process for generating Build Tickets for 3D printing of Assemblies is almost identical to the process for part files with a few small changes. When Build Tickets are to be generated for an assembly using the CAMWorks AM module, the following enhancement/changes will be visible in in the ***CAMWorks Additive Manufacturing*** user interface.

1. Part Selection Tab in Part Manager Dialog Box of Create AM Job Wizard

Parameters and options within the ***Part Selection*** tab will be applicable only when you are executing the AM Job for an assembly. This tab lists all the parts that comprise the assembly. Use this tab to select only those parts of the assembly that are to be considered for the AM Job. (Parts not be selected include hidden or suppressed parts, parts that don't fit within the machine volume and parts not aligned with the build plate.)
If you proceed with the settings in the dialog boxes of the ***Create AM Job Wizard*** without eliminating

those parts that do not fit within the machine volume or are not aligned with the Build Plate, then an error message indicating that Build Ticket cannot be generated will be displayed when you execute the Build command in the ***Build Processor*** dialog box.



Part Selection Tab of Part Manager Dialog Box in Create AM Job Wizard in AM User Interface

2. Disabling of Pattern Creation Functionality

The functionality of 3D printing multiple instances of a part model via the ***Create AM Job: Edit Pattern*** dialog box is available only for singular part files. It is not available for assembly files. The ***Edit Pattern*** dialog box will not be displayed in the ***Create AM Job Wizard*** when an assembly is selected.

3. 'Part Manager' and 'Part' Items in Additive Manufacturing Tree

Part Manager: This item will be displayed in the AM tree only after an AM Job has been executed for an assembly. It will not be displayed in the AM tree when the AM Job has been executed for a single part model. Double-clicking on this item displays the Part Selection tab of the *Create AM Job: Part Manager* dialog box. Use the options provided within this tab to select the parts within the active assembly file that are to be selected for additive manufacturing. (Select only those parts that fit within the machine volume and are aligned to the Build Plate.)

Part: This item will be displayed in the AM tree only after an AM Job has been executed for an assembly. Part items are listed under the *Part Manager* item in the AM tree represent those parts of the assembly that were considered for the AM Job.

The Support Faces will be listed under the *Part* item these support faces were created for.

In order to distinguish between the multiple parts of the assembly that were considered for the AM Job, a numerical suffix (in incremental values of '1') will be assigned to each Part listed under *Part Manager* item in the AM Tree.

Example:

Consider that an AM Job was executed for an assembly comprising 5 part models. Out of these 5 parts, only 3 were considered for the AM Job. In such a case, the Part items listed in the AM tree under Part Manager item will be *Part1*, *Part2* and *Part3*.

Tools

Improved - Additional Comments for Tools

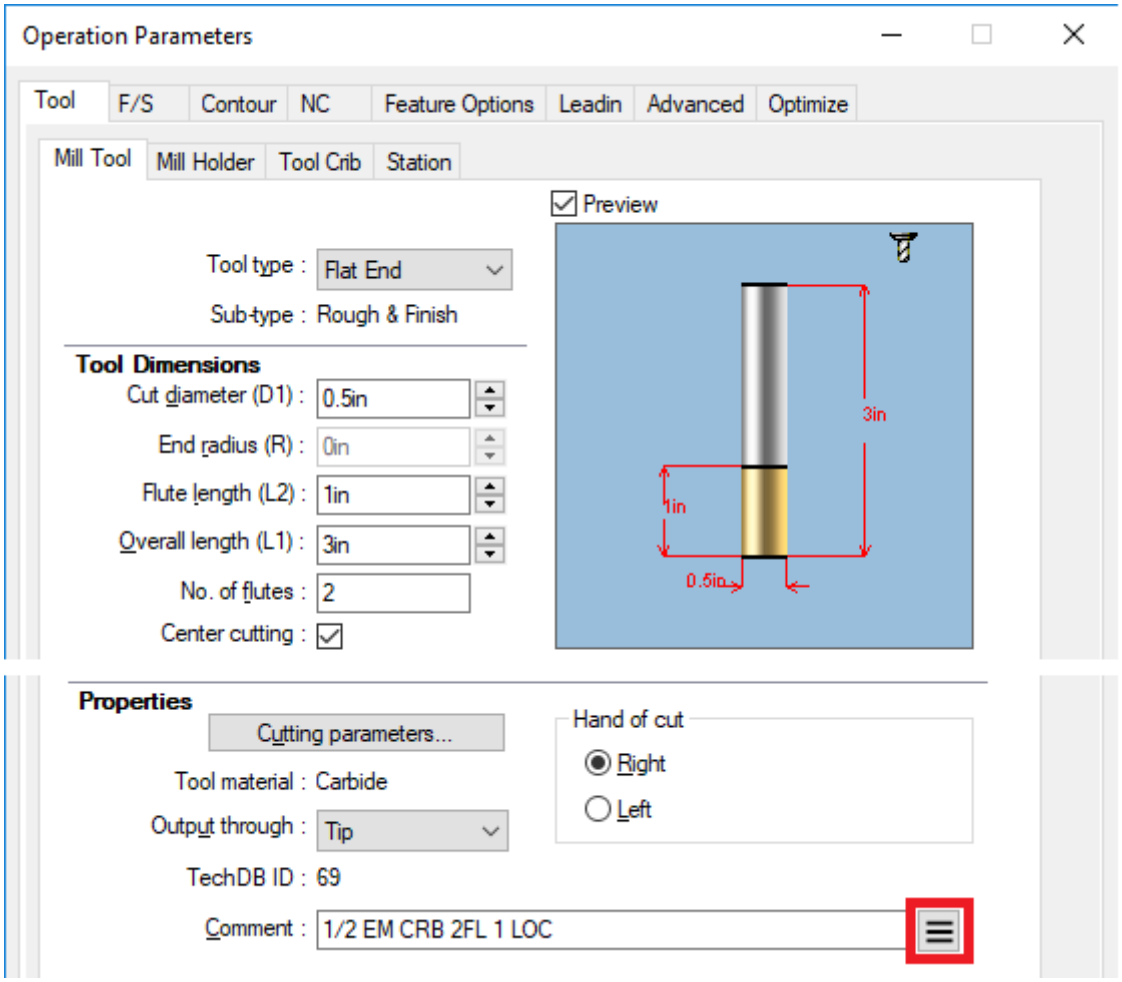
Purpose:

Enables all types of Mill tools, Turn Inserts, Tool Holders and Boring Bars to optionally have additional tool information (such as Tool ID, Vendor, Description) that can be output to the post processed G-Code and in Setup Sheets.

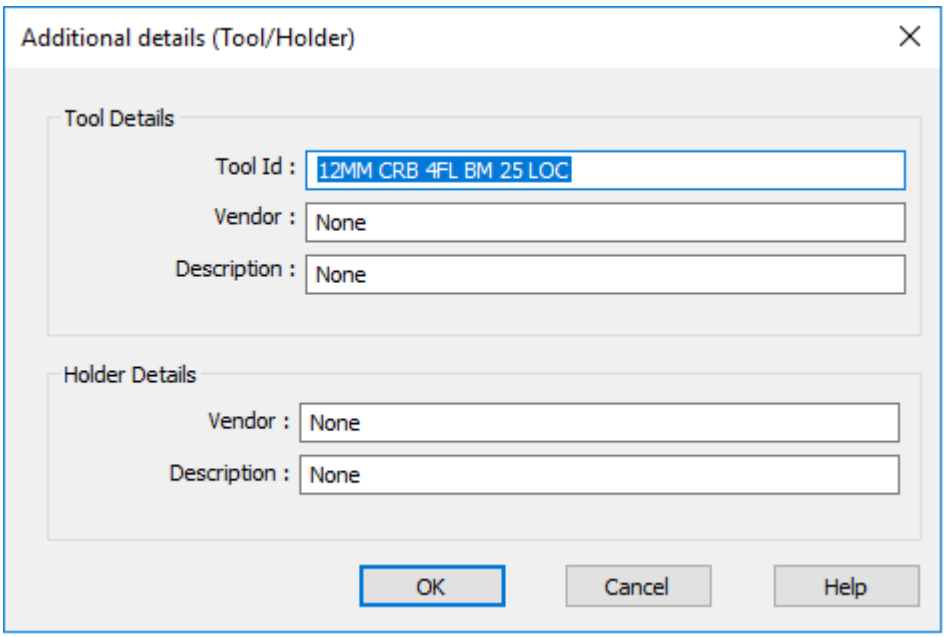
Implementation:

From ***CAMWorks 2020*** version onwards, in Mill mode, additional information (Tool ID, Vendor Description) pertaining to a mill tool and holder can be added using the ***Additional Details (Tool/Holder)*** dialog box. This dialog box is displayed when the following commands are executed:

- When you click on the ☰ button within the ***Properties*** group box in ***Tool*** page under the ***Tool*** tab of the ***Operation Parameters*** dialog box/ ***Edit Tool Parameters*** dialog box
- When you click on the ☰ button adjacent to the ***Comment*** field in ***Mill Holder*** page under the ***Tool*** tab of the ***Operation Parameters*** dialog box/ / ***Edit Tool Parameters*** dialog box
- When you click on the ☰ button adjacent to the ***Comment*** field in the ***New Tool*** dialog box (displayed when you click on the ***New Tool*** button under the ***Tool Crib*** tab of the ***Machine*** dialog box.



Addiitonal Details button in Tool Page underTool Tab of Operation

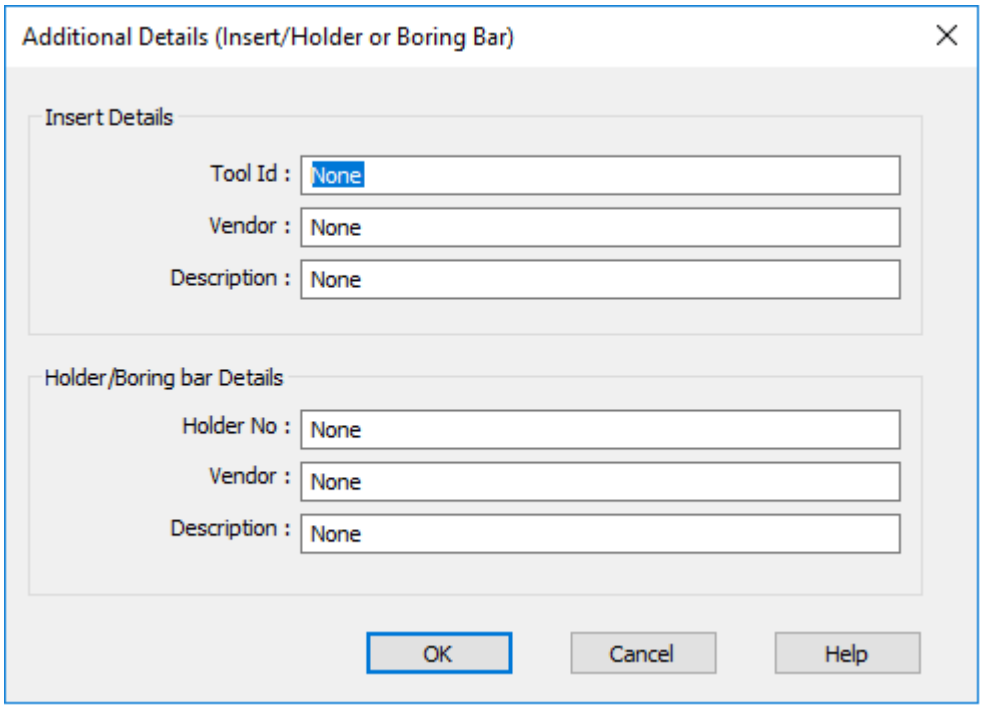


Additional Details (Tool/ Holder Dialog Box)

Turn Mode

In Turn mode, additional information (Tool ID, Holder Number, Vendor Description) pertaining to a Turn insert and Holder/ Boring Bar can be added using the ***Additional Details (Insert/ Holder or Boring Bar)*** dialog box. This dialog box is displayed when the following commands are executed:

- When you click on the  button within the adjacent to the ***Comment*** field in the ***Insert*** page under the ***Tool*** tab of the ***Operation Parameters*** dialog box/ ***Edit Tool Parameters*** dialog box
- When you click on the  button adjacent to the ***Comment*** field in ***Holder*** page/***Boring Bar*** page under the ***Tool*** tab of the ***Operation Parameters*** dialog box/ / ***Edit Tool Parameters*** dialog box



Additional Details (Insert/ Holder or Boring Bar Dialog Box)

Nature of the Information in the Additional Details (Tool/Holder) and Additional Details (Insert /Holder or Boring Bar) dialog boxes

- These dialog boxes can be used to view, assign and edit additional information about the cutting tool and holder (such as Tool ID, Vendor and Description).
- The values associated with the parameters in these dialog boxes can be alphanumeric in nature. These parametric values can be post processed as comments in the G-code.
- They will also be available in the Setup Sheets generated after post processing.
- The default values for these parameters will be retrieved and displayed from the TechDB. If no value has been defined for the parameter in the TechDB, the field associated with that parameter will be blank.

Saving Changes made to the Additional Information to TechDB

When you click on the OK button within these dialog boxes, the corresponding dialog box will close and any changes made to its parameters will be retained in the CAMWorks application only for that specific solid part/assembly. It won't be saved to the TechDB.

If you wish to save the changes made to the TechDB so as to peruse these values for future use, then following are the steps:

i. Open the ***Machine*** dialog box and click on ***Tool Crib*** tab.

ii. Click on the ***Save Tool Crib*** button within this tab to save the changes made to the tools within the active tool crib.

iii. The ***Save to Database*** dialog box will be displayed.

   a. To save the changes for the existing tool's entry in the TechDB, place a check in the Update Tool check box and click the ***Save*** button.

   b. To save the changes as a new tool entry in the TechDB without affecting the original tool entry, place a check in the ***Add as new tool*** check box and click the ***Save*** button.

iv. Clicking on the ***Save*** button saves the changes to the TechDB and closes the ***Save to Database*** dialog box.

Installation

New - MSI-Based Installer for CAMWorks Application to enable Command Line Support

Purpose:

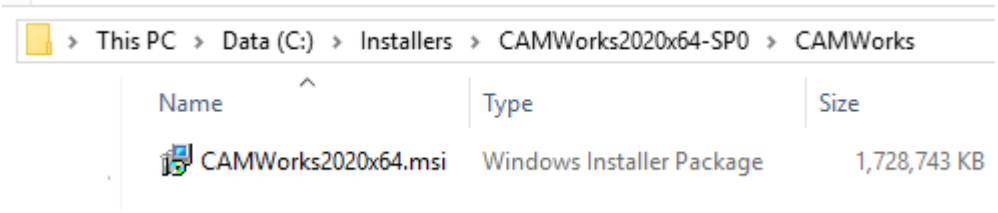
To make the CAMWorks Installer MSI-based in order to provide Command Line support and Push Support for remote installation/ silent installation by IT Administrators

Implementation:

During the manual installation process, the ***CAMWorks Installation Wizard*** will prompt (in the form of buttons and checkboxes) the user performing the installation to:

- Agree to the ***End User's License Agreement***,
- Indicate the folder where the application is to be installed,
- Opt in or out of the ***CAMWorks Feedback Program***,
- Select the languages to be installed,
- Select supplementary applications or utilities to be installed,
- Enable/disable performance feedback
- Indicate whether you wish to view the ***ReadMe*** file once the installation is complete

If your facility has multiple user-licenses of the CAMWorks application or a Floating Network license, then this installation/ upgrading activity needs to be performed on all the Windows machines on which the CAMWorks application is installed. The 'non-silent' mode of installation can prove to be cumbersome for your IT Administrator. This issue can easily be addressed by using a silent installer for CAMWorks.
From ***CAMWorks 2020*** version onwards, the functionality for creating a silent install process for the CAMWorks application is supported. From this version onwards, the installer for the CAMWorks application is an MSI-based installer.



MSI-based Installer for CAMWorks application in CAMWoks Folder of Installer Package

This installer is capable of taking inputs through the Windows Command Line prompt. A batch file for the silent installation process can be created using this MSI-based installer. Up on execution, the batch file will perform silent installation without the need for any manual intervention or clicking of any button within the ***Installation Wizard***.
An additional advantage of executing silent installation is that it enforces compliance to a specific version of CAMWorks across all machines on the network.

Note:

For details on the various variables, syntax and illustrative examples of command line arguments pertaining to CAMWorks Silent Installation, you may refer to ***Appendix B: Creating and Using CAMWorks Silent Installers*** of the ***CAMWorks_Installation_Guide.pdf*** document. This document is available in the ***CAMWorks Installer Package*** folder.

Posting

New - Enhanced Universal Post Generator (named "UPG-2")

Purpose:

Enhanced ***Universal Post Generator (UPG)*** installer named ***UPG-2*** with support for Probing Cycles and Subroutines

Implementation:

From *CAMWorks 2020* version onwards, the older versions of ***Universal Post Generator*** will no longer be supported. In its place, an enhanced version of UPG named ***UPG-2*** will be available.
UPG-2 is built using technologies that are compatible with the modern Operating Systems. It has the same structure, concepts, and methodologies as the old UPG. Users will not find any difference in the user interface of this application.

- Existing SRC or LIB files will get compiled as before.
- Post processors compiled using *UPG-2* will be compatible with older CAMWorks versions (previous to *CAMWorks 2020*).
- For better compatibility, *UPG-2* gets installed as a separate installation and will not overwrite an older version of UPG.

UPG-2 also comes equipped with the enhancements required to support Probing and Subroutines in ***CAMWorks 2020*** version.
Please refer the document ***Whats_New_in_UPG_2.pdf*** available with the link for downloading the ***UPG-2*** installer for details on all these newly introduced commands, constants, CALC sections and variables. Alternatively, after UPG-2 is installed, this document can be viewed using its Windows shortcut menu (*Start>>Programs>>UPG-2>>What's New in Universal Post Generator*).