CAMWorks®
A Geometric Product

What's New in CAMWorks 2015

What's New in CAMWorks 2015

Geometric

April, 2015

Table of Contents

What's New in CAMWorks 2015 – SP1.1

Updated - 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 (SP1.1).
Implementation:	To view the document, select: *Start>>All Programs>>CAMWorks2015x64>>Resolved CPR's.*

What's New in CAMWorks 2015 – SP1

Updated - 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 (SP1).
Implementation:	To view the document, select: *Start>>All Programs>>CAMWorks2015x64>>Resolved CPR's.*

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler	The 64-bit versions of : - SOLIDWORKS 2014 - CAMWorks Solids 2014 - SOLIDWORKS 2015 - CAMWorks Solids 2015
Operating System	64-bit version of Windows 8.1 and Windows 7 (SP1 or higher). [*Home Editions are not supported] **Note:** CAMWorks 2015 is not supported on 32-bit systems.

Improved - New strategy for Multi Surface Feature conditions

Purpose:	New strategy for Multi Surface Feature conditions that require undercut machining.
Implementation:	In CAMWorks 2015 SP1, a new strategy has been provided for the multi surface features. This strategy is named as "3 Axis-Undercut". The operation parameters under this strategy have been set to machine a Multi Surface Feature which requires undercut machining on a 3 axis Milling machine. **Note:** This strategy will be available only to those customers who use the default TechDB installed by CAMWorks.

Improved - MS Access 2013 and MS Office 365 support for Technology Database

Implementation:	The CAMWorks application requires Microsoft Access for the functioning of its TechDB (Technology Database™). The previous versions of CAMWorks 2015 application supported the MS Access versions of Access 2003, 2007, 2010 and 2010 SP1. From *CAMWorks 2015 SP1* version, in addition to the existing versions, the following versions of MS Access too will be supported for functioning of TechDB: • MS Access 2013 SP1 • MS Office 365 with Office 2013

Improved - Provision to borrow license for 30 days on a floating network

Purpose:	- To increase the number of days for which a floating license can be borrowed from 7 days to 30 days. - To allow non-admin users to borrow floating licenses.
Implementation:	The number of days for which a floating license can be borrowed is embedded in the license file. Up to CAMWorks 2015 SP0, license files conferred to all customers allowed floating licenses to be borrowed only by users with admin rights and the borrow period was limited to maximum 7 days. From CAMWorks 2015 SP1 onwards, the Floating License Borrowing feature has been upgraded as follows: - Floating licenses can be borrowed by user with admin rights as well as users without admin rights. - Floating licenses can be borrowed for up to 30 days replacing the 7-day maximum limit. **Note:** - Enabling of the above features is dependent on the CAMWorks license file since these features are embedded in the license file code. All fresh CAMWorks license files issued to customers by Geometric after the release of CAMWorks 2015 SP1 will have the upgraded license borrowing feature enabled. - If you already have a CAMWorks floating license and an active Update Support Plan, then the upgraded license borrowing feature will be activated when you renew your CAMWorks license or Update Support Plan. If you wish to avail this upgraded functionality at any point of time before your CAMWorks license/ Update Support Plan expires, then please contact your CAMWorks reseller.

Improved -Side Allowance enabled for Contour Mill operations of Curve Features

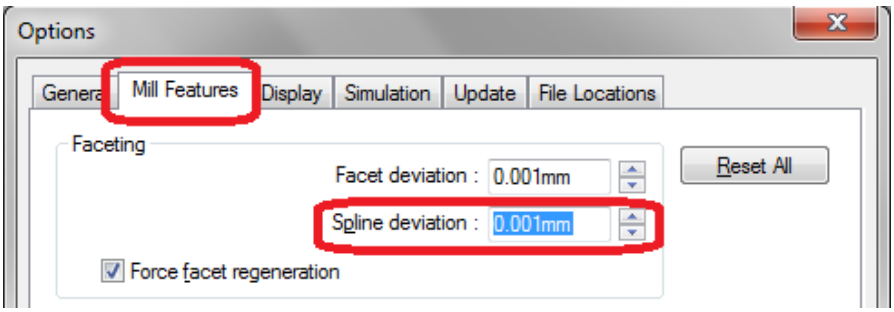
Purpose:	Allows users to assign Side Allowance parameters for Contour Mill operations defined for Curve and Engrave features when Toolpath Center is set to *With Compensation*.
Implementation:	In previous versions of CAMWorks, the Side Parameters Allowance parameter was disabled for Contour Mill operation defined for Curve features or Engrave features when the *Toolpath center* (in NC tab) was set to *With Compensation*. As a result, users couldn't assign a Side Parameter Allowance which would then be considered in the Contour Mill Toolpath calculation. From CAMWorks 2015 SP1, this functionality has been enabled.

New - APIs introduced in CAMWorks 2015

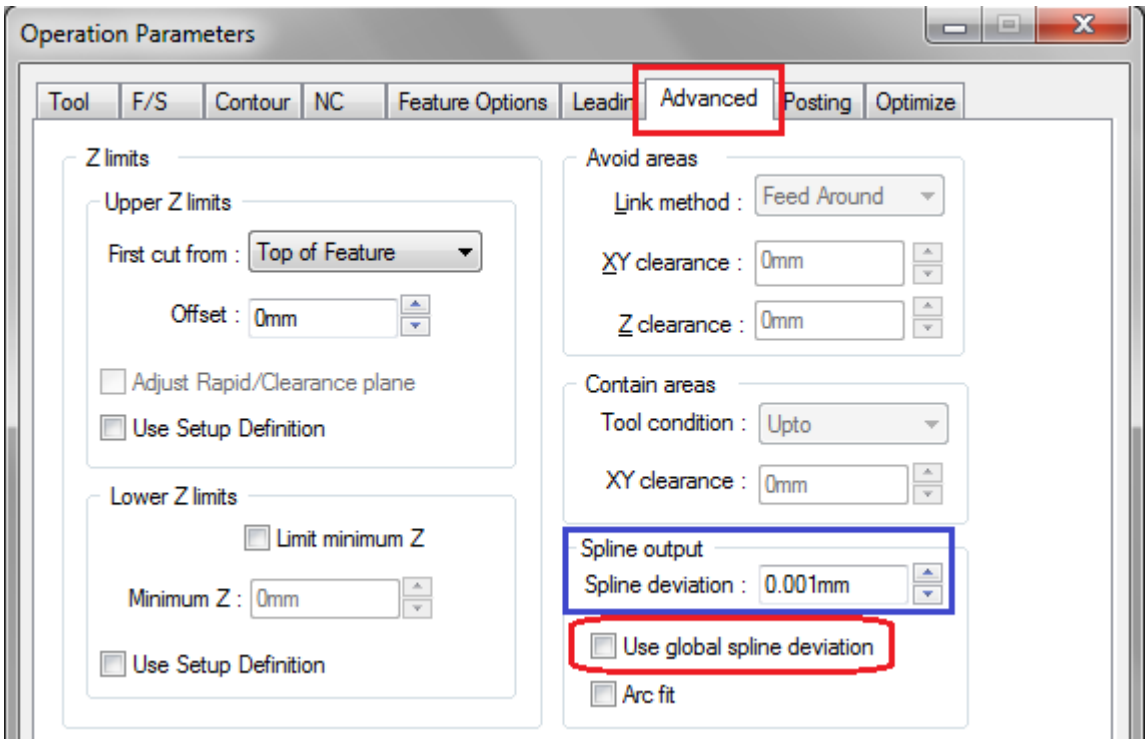
Purpose:	New document listing all the APIs available in CAMWorks
Tutorial	To view the list of APIs provided in CAMWorks, along with sample macros provided for APIs introduced in CAMWorks 2015, click on the Windows Start menu and select ***All Programs>>CAMWorks 2015>>Manuals>>APIs available in CAMWorks***.

Improved - Improved toolpaths for Engrave Features with Spline Deviations

Purpose:	For engrave feature created using splines: - Users can now assign Spline Deviation for Contour Mill operations defined for Engrave Feature from within the operation parameters - Better quality toolpaths
Implementation:	In previous versions of CAMWorks: - The Spline Deviation for Contour mill operation machining the Engrave features could only be set globally from the *CAMWorks Options* dialog box. The provision to assign a Spline Deviation value locally for such Contour mill operations was not available. - For Engrave features with tighter spline deviations, the time required for machining the toolpath was more since the toolpath consisted of multiple jumps and lifts. From CAMWorks 2015 SP1 version onwards: - Users can modify/change the Spline Deviation value for Contour mill operations machining an Engrave Feature. This parameter is present on the *Advanced* tab of the Operation Parameters dialog box. Unchecking this option enables the ***Spline Deviation*** parameter. - Toolpaths for engrave features with tighter spline deviations have fewer jumps and lifts resulting in lesser time consumed for machining the toolpaths as well as better quality toolpaths.



Spline Deviation Defined Globally in CAMWorks Options dialog box

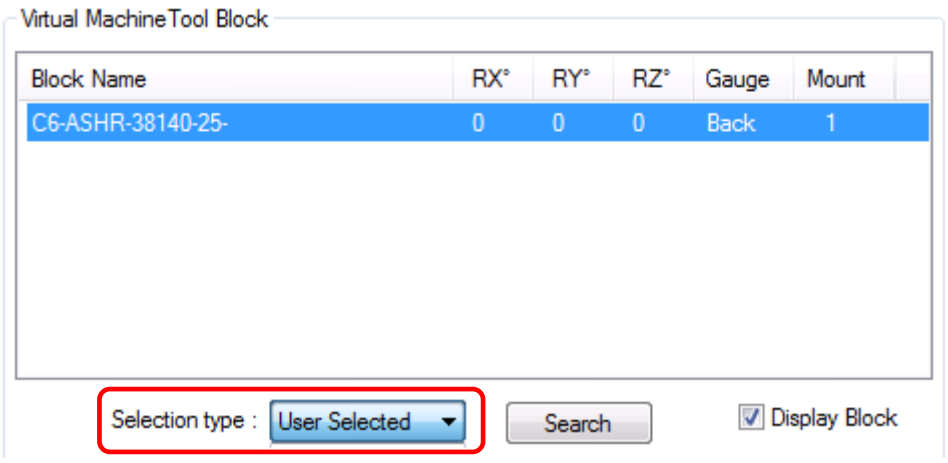


Assigning Spline Deviation locally in advanced Tab of the Operation for Engrave Feature

Improved -Tutorials and Sample Parts for the CAMWorks Virtual Machine module

Purpose:	- Improvised Tutorials provided for the CAMWorks Virtual Machine application. - Both partially programmed and fully programmed Sample Part files provided for reference.
Implementation:	**Three separate Tutorial documents have been provided:** - CAMWorks Virtual Machine Tutorial for Mill - CAMWorks Virtual Machine Tutorial for Turn - CAMWorks Virtual Machine Tutorial for Mill-Turn **Salient Features of the documents:** - Step-by-step details on how to execute Machine Simulation using the CAMWorks Virtual Machine module provide in the form of tutorials. - Smart illustrations and pictorial representations to enable better understanding of various functionalities, features and settings. - Explanation of the various commands and settings provided in the CAMWorks Virtual Machine user interface. **Partially Programmed and Fully Programmed Sample Part files:** - Partially Programmed sample parts and assemblies have been provided for executing the tutorials given in the CAMWorks Virtual Machine Tutorial documents. - For Fully Programmed sample parts and assemblies, the CAMWorks parametric settings have already been assigned. These parts/ assemblies can be used to directly simulate toolpaths in CAMWorks Virtual Machine.
Tutorial Location:	To acess these documents from the Windows *Start* menu, select ***All Programs>>CAMWorks 2015x64>> Manuals>>CAMWorks Virtual Machine*** and select the applicable tutorial document from the menu.
Sample Parts Location:	Samples parts/assemblies are provided at the following location: Drive: \CAMWorksData\CAMWorks2015x64\MachSim\Samples Within this folder location, the sample parts and assemblies are classified as per machine names (*Mill/Turn/Mill-Turn*). These Machine folders are further classified as Partially and Fully programmed sample parts/assemblies. For example, sample parts/assemblies for Mill Machine Simualtion are provided at the following locations: • Drive: \CAMWorksData\CAMWorks2015x64\MachSim\Samples\Mill\ Partially_Programmed • Drive: \CAMWorksData\CAMWorks2015x64\MachSim\Samples\Mill\ Fully_Programmed

New - User Defined Tool Blocks for CAMWorks Virtual Machine

Pre-requisite:	You need the CAMWorks Virtual Machine License to access this functionality
Purpose:	Allows an STL file accompanied with a Tool Block Definition file to be saved as a User Defined Tool Block. A Tool Block thus defined can be selected as a Tool Block during Machine simulation for Turn and Mill-Turn Machines.
Implementation:	**Creating a User Defined Tool block** The ***User Defined Tool Blocks*** command on the CAMWorks menu allows you to save an STL file accompanied with a data file containing tool block information to be saved as a Tool Block within SOLIDWORKS/CAMWorks Solids. Such a paired STL file and Tool Block Definition file (with extension **.TBDINF*) will be stored in the *Tool Block* folder of the Virtual Machine associated with the Tool Block being created. The paired file can be loaded into CAMWorks Turn and Mill Turn modes as accurate representations of Tool Blocks that are used in Virtual Machine Simulations for collision detection. **Note:** - In addition to creating a Tool block from a model, the tool block data may be also be created and edited independent from the Solid model. - Tool Blocks can be defined from any coordinate system as CAMWorks will calculate the required rotation needed to mount the block in the Turret. **Selecting a User Defined Tool block** The ***Virtual Machine Tool Block*** group box located in the Station page of the Tool tab of Operation Parameters dialog box allows for User Selected Tool Blocks to be viewed/selected/changed/removed. When the Selection Type within this group box is set to ***User Selected***, *CAMWorks* allows a block to be selected for the active tool. The selected block and tool will be used for all Virtual Machine simulations until the selection is changed or removed. If the active machine is changed to a machine that does not support the selected block, then CAMWorks will revert to ***Automatic*** selection during machine simulation sessions until a new block supported by the active machine is selected. All selections are based on the active tool and active Virtual Machine.
	
Tutorials:	For detailed information on how to create/edit User Defined Tool Blocks, please refer the Chapter titled ***Creating and Using User Defined Tool Blocks*** in the *CAMWorks Virtual Machine Tutorial documents* for Turn and Mill-Turn. To acess these documents from the Windows Start menu, select *All Programs>>CAMWorks 2015x64>> Manuals>>CAMWorks Virtual Machine*.

New - New Posting Variables

New Query	Purpose	Syntax
QUERY_FEATURE_STRATEGY	If the return value of QUERY_RESULT is TRUE, the system will pass the feature strategy in the post variable called QUERY_CHAR_VAL.	QUERY_ITEM_ID=QUERY_FEATURE_STRATEGY
QUERY_USE_SUB_PROGRAMS	If the return value of QUERY_RESULT is TRUE, then the system will pass 1 or 0 if in assembly mode irrespective of whether the Subprogram checkbox is checked or not. The value will be stored in the post system variable called QUERY_INT_VAL.	QUERY_ITEM_ID=QUERY_USE_SUB_PROGRAMS

New System Variables	Type	Usage
OPR_REVERSE_ARC_DIR	DECIMAL Detects if the Mill operations OD or FACE arcs need to be reversed or not in a Mill-Turn post.	OPR_REVERSE_ARC_DIR=0 - No reversal is needed. OPR_REVERSE_ARC_DIR=1 - The arc directions should be reversed.
OPR_Z_DEPTH	METRIC DECIMAL	This is used for Drilling only and not available for Milling.
O_CYCLE_START_X	DECIMAL	Passes the X start point of Turn Canned OD, ID and Face cycle. This value will be used before the canned cycle is output.
O_CYCLE_START_Z	DECIMAL	Passes the Z start point of Turn Canned OD, ID and Face cycle. This value will be used before the canned cycle is output.

New Post Variable	Type	Usage
IS_PATTERN_FEATURE	INTEGER	Passes information to the post whether the feature is patterned or not. The value will either be TRUE or FALSE.
IS_TLP_COMPENSATED	INTEGER	Passes information to the post whether the toolpath is compensated or not. The value will either be TRUE or FALSE.
STOCK_MAX_X	DECIMAL	Stores the Stock Maximum value in X axis.
STOCK_MAX_Y	DECIMAL	Stores the Stock Maximum value in Y axis.
STOCK_MAX_Z	DECIMAL	Stores the Stock Maximum value in Z axis.
STOCK_MIN_X	DECIMAL	Stores the Stock Minimum value in X axis.
STOCK_MIN_Y	DECIMAL	Stores the Stock Minimum value in Y axis.
STOCK_MIN_Z	DECIMAL	Stores the Stock Minimum value in Z axis.
5AXIS_FEED_DISTANCE	DECIMAL	Can be used instead of DISTANCE, N_DISTANCE and P_DISTANCE.
USE_FEATURE_OD_ID	INTEGER	This variable can be set in a Turn or Mill-Turn post at either CALC_START_OF_TAPE in your post library file or CALC_INIT_CODES in the SRC file.

New System Commands	Purpose	Syntax
GETMCS	To get all the different MCS offsets inserted in the current part file using ProCAM II or CAMWorks.	GETMCS(1,calc_section) GETMCS(3,calc_section) **Note:** • Using GETMCS (1, calc_section) will call all Setups, including duplicate Setups. • Using GETMCS (3, calc_section) will call all unique Setups with different angles, thus excluding duplicate Setups.

What's New in CAMWorks 2015 – SP0.1

Updated - 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 (SP0.1).
Implementation:	To view the document, select: *Start>>All Programs>>CAMWorks2015x64>>Resolved CPR's.*

What's New in CAMWorks 2015 – SP0

Updated - 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 (SP0).
Implementation:	To view the document, select: *Start>>All Programs>>CAMWorks2015x64>>Resolved CPR's.*

Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler	The 64-bit version of SOLIDWORKS 2014 and CAMWorks Solids 2014. **Note:** CAMWorks 2015 has been tested to work with the latest available release of SOLIDWORKS 2015 (Pre-Release).
Operating System	64-bit version of Windows 8.1 and Windows 7 (SP1 or higher). [*Home Editions are not supported] **Note:** CAMWorks 2015 is not supported on 32-bit systems.

General

New - Assigning solid model as stock in Stock Manager Dialog box

Purpose:
Enables selection of a solid body as stock for the part being machined.

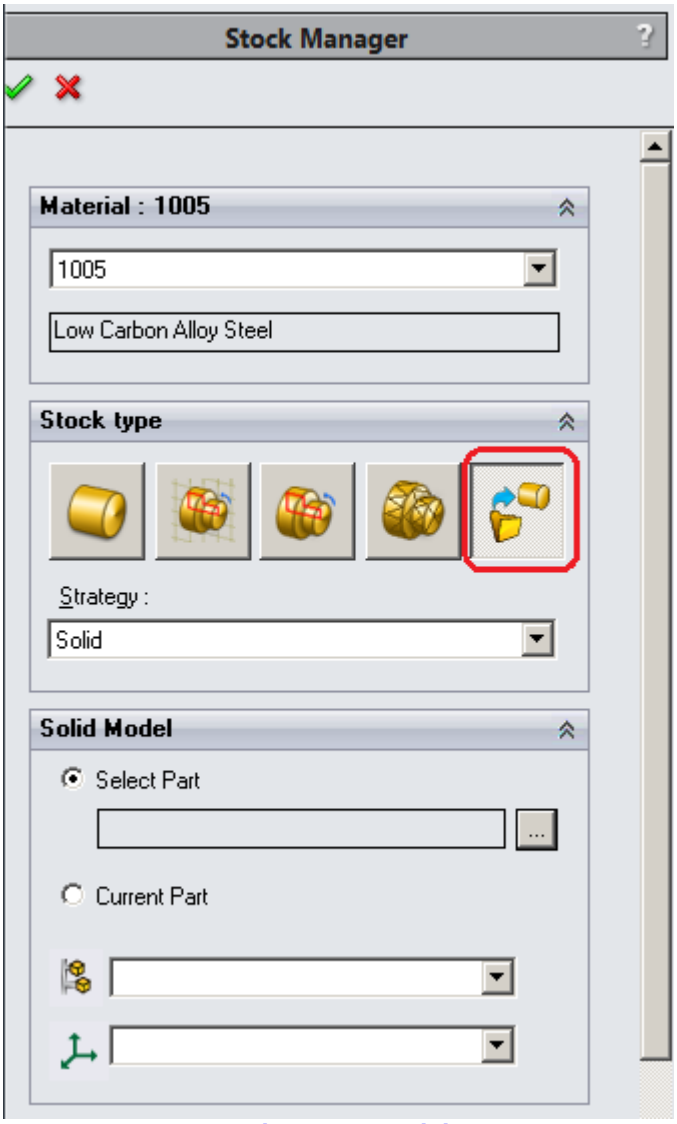
Implementation:
In previous versions of CAMWorks, the functionality to select a solid body as stock was available only in Mill Assembly mode.
From CAMWorks 2015 version onwards, this functionality has been extended to Stock Manager dialog box of Mill, Turn, Mill-Turn and Wire EDM.

You can assign either a desired SOLIDWORKS part file or another SOLIDWORKS configuration from the current part file as the stock.

To define the stock from a solid model:

1. Select Part file as the stock type.
2. To assign another part file as the stock,
   a) Click on the *Select Part* option.
   b) Click on the *Browse for part file* button  and select the part file.
   c) If the selected part file contains multiple SOLIDWORKS configurations, select the desired configuration from the SOLIDWORKS Configurations dropdown list  .
3. To select another SOLIDWORKS configuration as the stock:
   a) Select the *Current Part* option.
   b) Select the desired configuration from the SOLIDWORKS Configurations dropdown list  .

Note:
If the selected SOLIDWORKS Configurations contains multiple bodies, then all the bodies will be considered as stock.



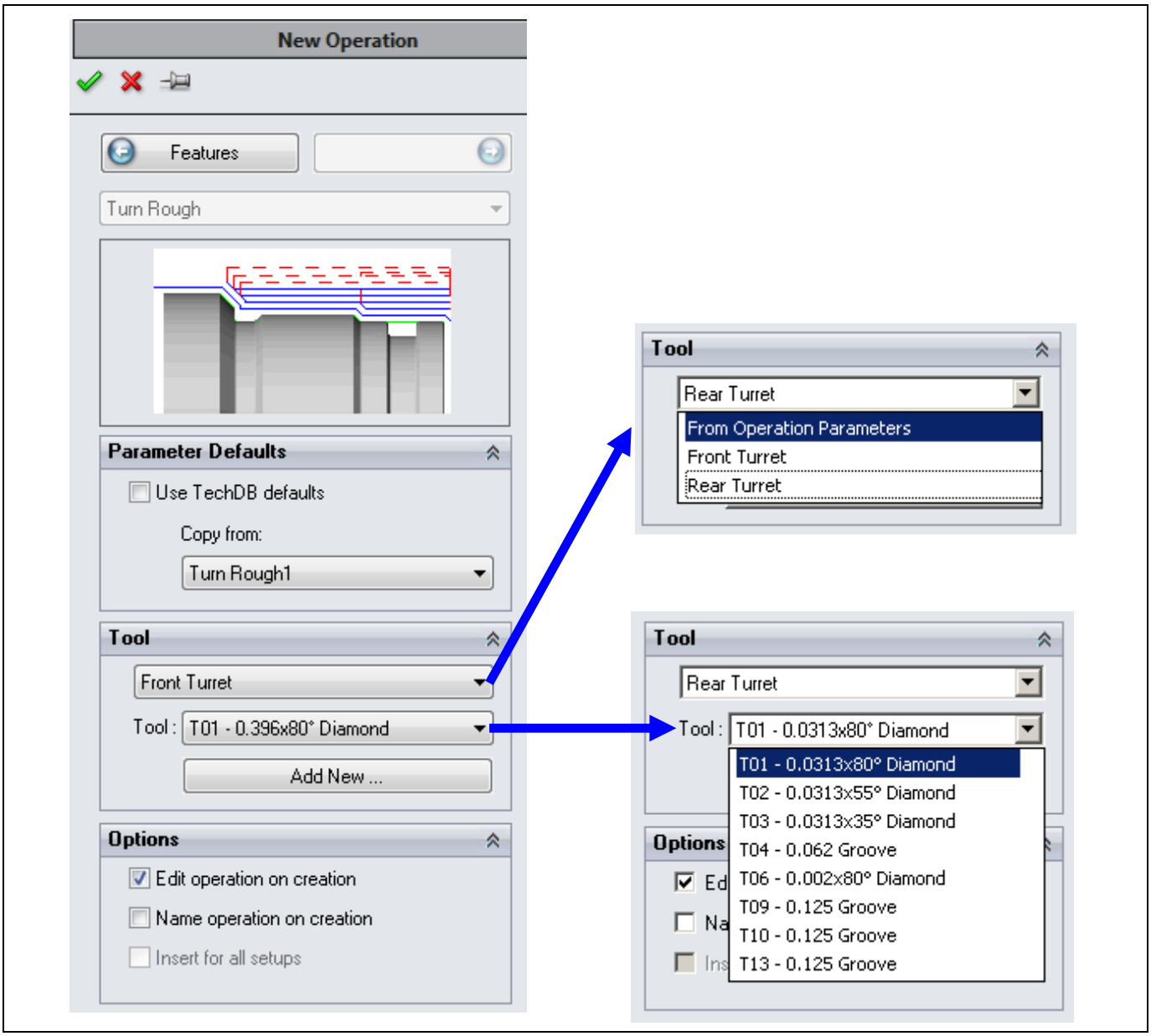
Turn Stock Manager Dialog Box

New - Posting Tab in Part Setup dialog box and Operation Setup dialog box

Purpose:	Parameters in the Posting tab of the Part Setup dialog box and Operation Setup dialog box provide the ability to define posting parameters which cannot be fulfilled by toolpaths.
Implementation:	In previous versions of CAMWorks, posting parameters were assigned in the Posting tab of the Machine dialog box. These include posting parameters such as movement of stock and tailstock on Turn machines, assigning safe start value for operations, etc. From CAMWorks 2015 onwards, these parameters can also be assigned in the Posting tab of: • Part Setup Parameters dialog box when a Mill part, Mill Assembly or Mill-Turn part. • Operation Setup Parameters dialog box when machining a Turn part. **Note:** The parameters displayed in this tab will depend on the post processor selected.

New - Provision to assign tool to operation in New Operation Dialog Box

Purpose:	Provides an option within the New Operation dialog box to assign the tool to the Mill, Turn or Hole operation being interactively inserted.
Implementation:	The option to assign the tool to the operation being interactively inserted is available in the second page of the New Operation dialog box. • When a Mill machine is selected, this option allows you to select whether the tool (which will be assigned to the Mill operation) will be picked from a referenced Operation or from the Tool crib. • When a Turn or Mill-Turn machine is selected, this option allows you to select whether the tool (which will be assigned to the Mill or Turn operation) will be picked from a referenced Operation or from the Front/Rear Turret. If the suitable tool is not available in the Tool Crib/Front Turret/Rear Turret, then the desired tool can be added from the Tool Library and assigned to the operation. Once the tool is assigned to the operation, a Preview window containing the dynamic 3D model view of the tool is displayed, thus enabling visual identification of the selected tool. You can zoom, pan and rotate the displayed 3D model view. The Tool Station number, Tool Comment and labels of the Tool parameters are also displayed in the Preview Window.

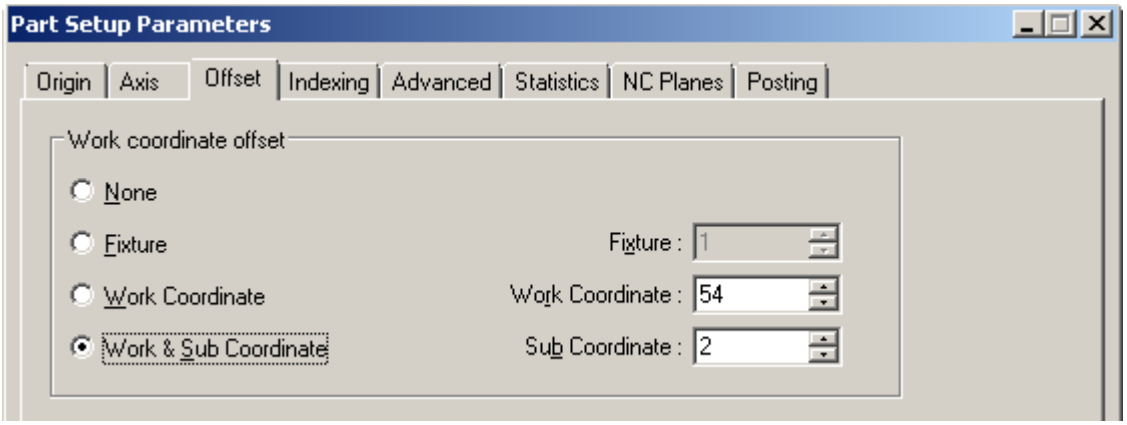

New Operation
Features
Turn Rough
Parameter Defaults
Use TechDB defaults
Copy from:
Turn Rough1
Tool
Front Turret
Tool : T01 - 0.396x80° Diamond
Add New ...
Options
Edit operation on creation
Name operation on creation
Insert for all setups
Tool
Rear Turret
From Operation Parameters
Front Turret
Rear Turret
Tool
Rear Turret
Tool : T01 - 0.0313x80° Diamond
T01 - 0.0313x80° Diamond
T02 - 0.0313x55° Diamond
T03 - 0.0313x35° Diamond
T04 - 0.062 Groove
T06 - 0.002x80° Diamond
T09 - 0.125 Groove
T10 - 0.125 Groove
T13 - 0.125 Groove
Options

New - Tutorial on Setup Sheets

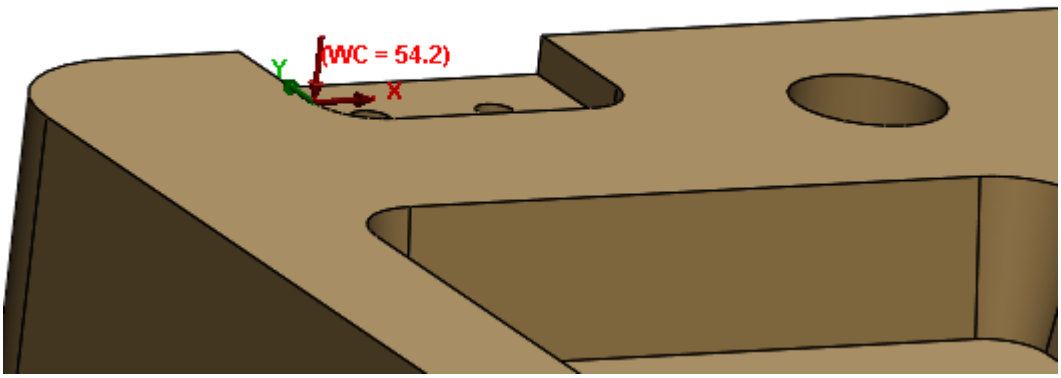
Purpose:	Serves as an effective guide for understanding how to generate and view Setup Sheets as well as how to customize/create Style Sheet templates.
Implementation:	The Setup Sheets Tutorial can be accessed from the Windows Start menu by selecting All Programs>>CAMWorks 2015x64>>Manuals>>Setup Sheets Tutorial. This document covers the following topics: • Generating XML-based and Access-based Setup Sheets • Nomenclature of generated XML-based Setup Sheets • Style Sheet Templates for XML-based Setup Sheets • Differences between Setup Sheet command at NC Manager Level and Setup level • Settings for Setup Sheets in CAMWorks Options Dialog box • Settings available in the Setup Sheet Options dialog box • Tutorials on Generating Setup Sheets • Viewing Saved Setup sheets • Customizing/Creating/Editing Style Sheet Templates

New - Display of Work Coordinates for a Setup in the graphics area

Purpose:	Provides the ability to view Work Coordinates of the Mill Setup or Turn Setup in the graphics area
Implementation:	In previous versions of CAMWorks, the Work Coordinates (and Sub Coordinates, if applicable) could be viewed only in the Offset tab of the Part Setup Parameters dialog box (for Mill and Mill-Turn parts) and Operation Setup Parameters dialog box (for Turn parts). From CAMWorks 2015 version onwards, the Work Coordinates and Sub Coordinates will be visible in the graphics area whenever the Mill or Turn Setup is highlighted in the CAMWorks trees.



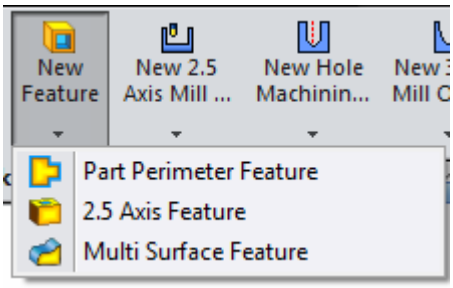
Offset tab of Part Setup Parameters dialog box



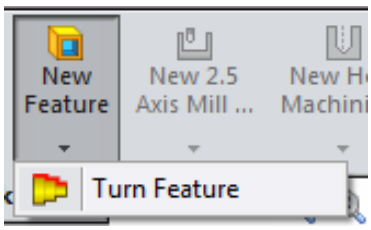
Work Coordinates displayed in Graphics area when Setup is highlighted in the CAMWorks Feature tree

New - Introduction of 'New Feature' button on the CAMWorks Command Manager

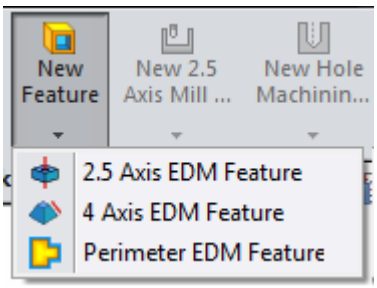
Purpose:	Provides a button command in the CAMWorks Command Manager/ Workflow toolbar to interactively insert features.
Implementation:	In earlier versions of CAMWorks, the command to interactively insert a new feature in available • In the **RMB context menu** of the Mill Part Setup and Feature node items in the CAMWorks Feature tree • From the CAMWorks pull-down menu (present in the SOLIDWORKS menu bar area) by selecting ***New>>Feature>>Feature Type*** from the cascading menu. The CAMWorks Command Manager/ Workflow toolbar provides the easiest route for executing CAMWorks commands using just one or two left-mouse clicks. From CAMWorks 2015 version onwards, a 'New Feature' button command has been introduced in the CAMWorks Command Manager and Workflow toolbar to enable interactive insertion of features. When this button is clicked, a dropdown list of features which can be interactively inserted is displayed. The user needs to click on the desired feature in order to view the dialog box required for interactively inserting the specific feature. The **New Feature** button is Machine sensitive i.e. the features displayed in the dropdown list on clicking this command button depends on the Machine type currently selected. • If a Mill machine is selected, then the dropdown list will display Part Perimeter Feature, 2.5 Axis Feature and Multi Surface Feature. • If a Turn machine is selected, then the dropdown list will display only Turn Feature. • If a Wire EDM machine is selected, then the dropdown list will display 2.5 Axis EDM Feature, 4 Axis EDM Feature and Perimeter EDM Feature.



New Feature dropdown list when Mill Machine is selected



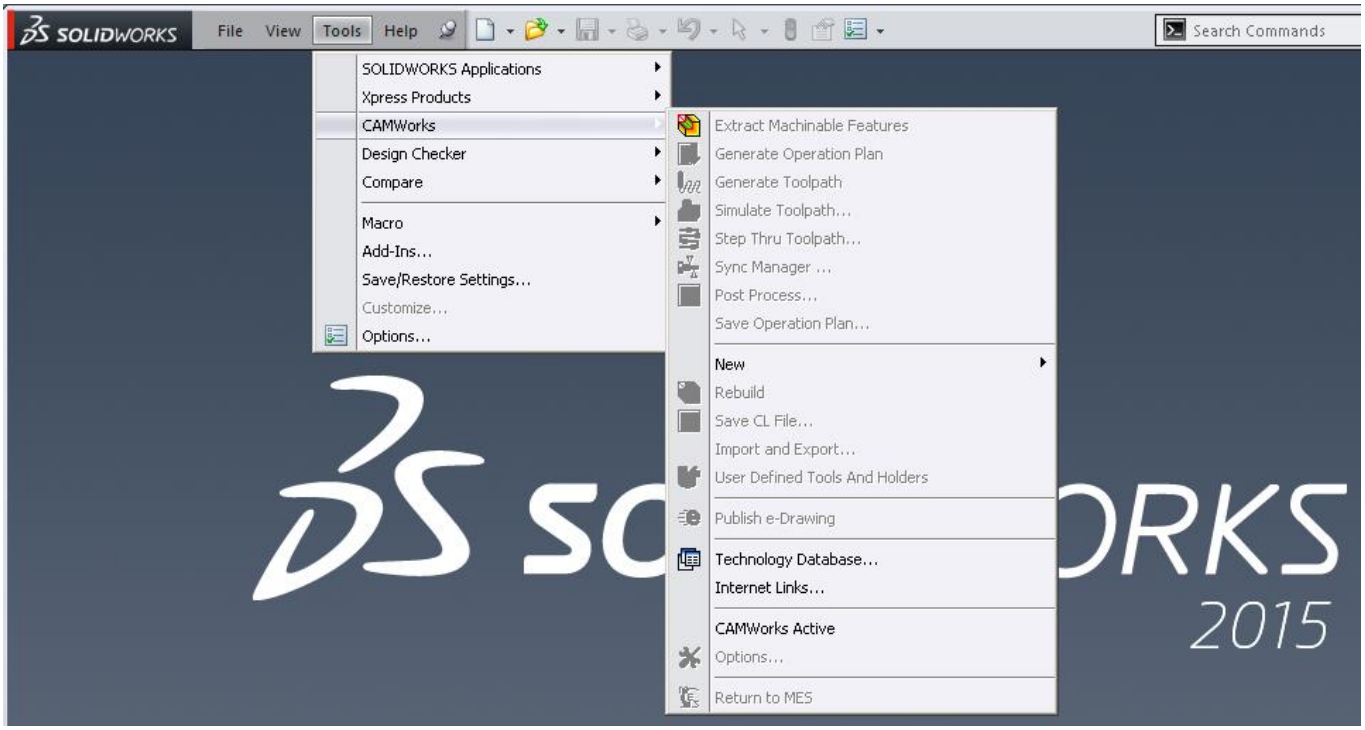
New Feature dropdown list when Turn Machine is selected



New Feature dropdown list when Wire EDM Machine is selected

Updated - Change in location of CAMWorks menu in SOLIDWORKS 2015

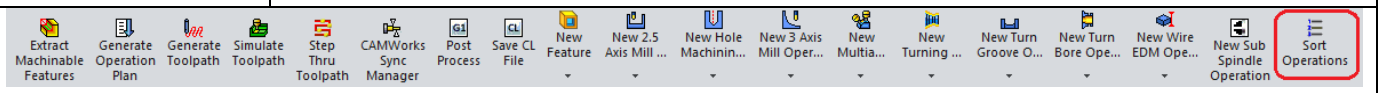
Purpose:	Change in location of CAMWorks menu in SOLIWORKS 2015/ CAMWorks Solids 2015
Implementation:	In SOLIDWORKS 2014/ CAMWorks Solids 2014 and previous versions, the CAMWorks menu was available in the SOLIDWORKS menu bar. From the SOLIDWORKS/CAMWorks Solids 2015 version onwards, the CAMWorks menu will be available as a cascading menu item under the "Tools" menu of the SOLIDWORKS menu bar.



CAMWorks menu available as a cascading menu item under Tools menu of SOLIDWORKS menu bar

New - Sort Operations command button on the CAMWorks Command Manager

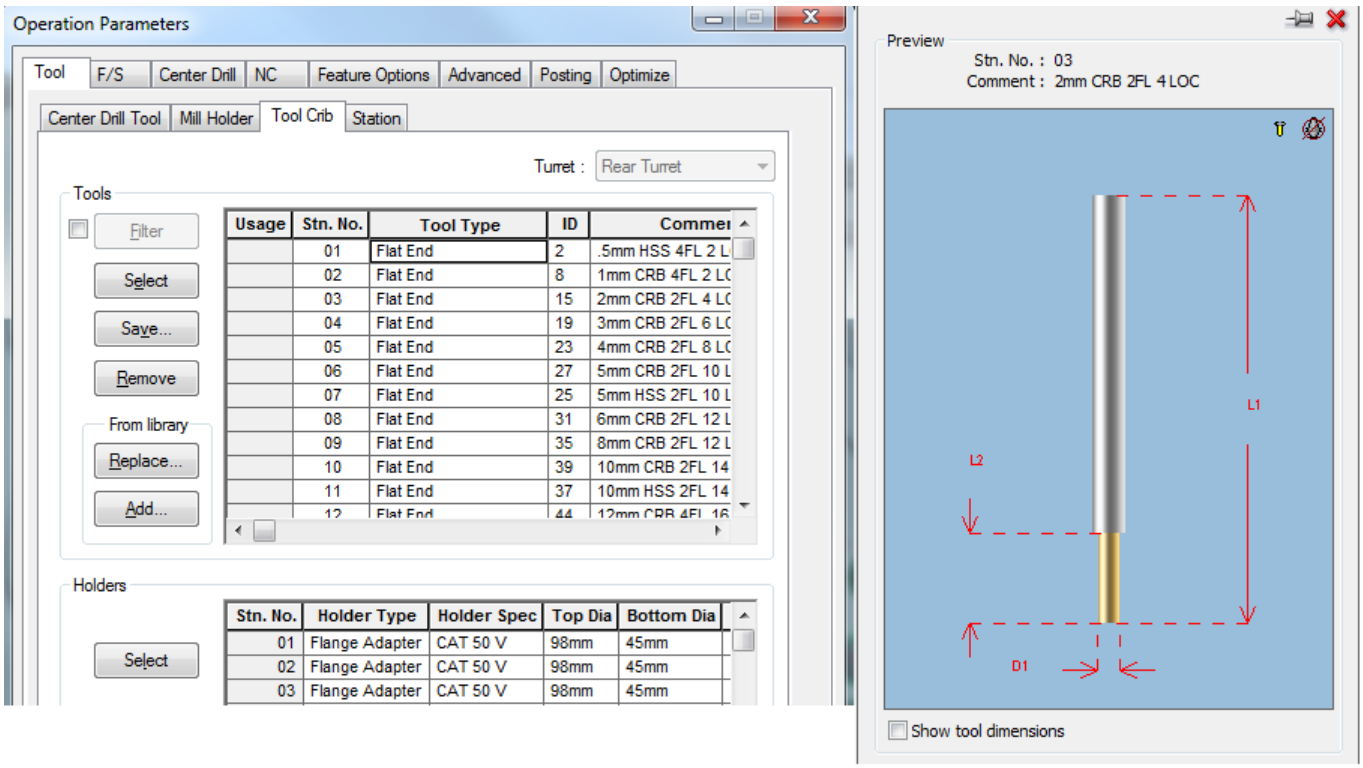
Purpose:	Availability of 'Sort Operations' command on CAMWorks Command Manager and CAMWorks Workflow toolbar
Implementation:	In previous versions of CAMWorks, the Sort Operations command, which allows you to sort operations in a logical machining sequence based on user-defined sort rules, was available for execution only from the following locations: • The CAMWorks menu • The context menu of the CAMWorks NC Manager item in the CAMWorks Operation tree • The context menu of the Mill Part Setup in the CAMWorks Operation tree From CAMWorks 2015 version onwards, the 'Sort Operations' command has been made available on the CAMWorks Command Manager and CAMWorks Workflow toolbar for easier access.



Sort Operations command on the CAMWorks Command Manager

New - Dynamic Preview of tool in Tool Crib Page of Operation Parameters dialog box

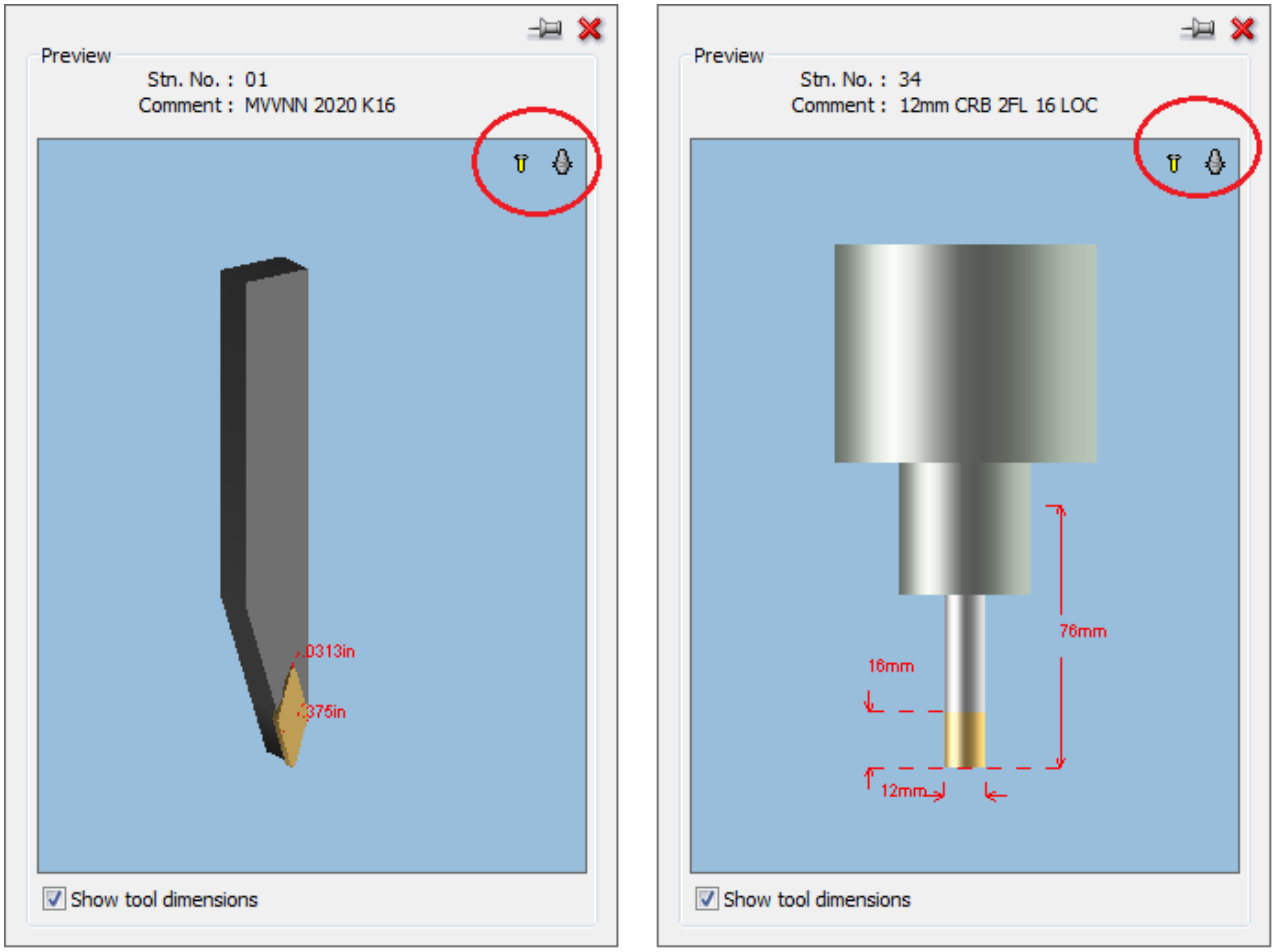
Purpose:	Provides a dynamic preview of the selected tool in the Tool Crib page under Tool tab of the Operation Parameters dialog box. The 3D model view of the tool in the Preview window enables visual identification of the tool.
Implementation:	From CAMWorks 2015 version onwards, a Preview window will be displayed when a tool is selected in the Tool Crib page under Tool tab of the Operation Parameters dialog box. This Preview window contains the dynamic 3D model view of the tool. The Tool Station number, Tool Comment and labels of the Tool parameters are displayed in the Preview Window. Users can zoom/pan/rotate this tool model to the desired orientation. - Use the scroll button of the mouse to zoom in/out the tool view. - Press the scroll button to rotate the tool view. - Use the right mouse button to pan the tool view. - Double click within the Preview Window to reset the tool view to default orientation.



Preview Window displaying 3D model of selected Tool when a Tool is selected in Operation Parameters dialog box

Improved - Display options in Preview window containing 3D Model view of Tool and Holder

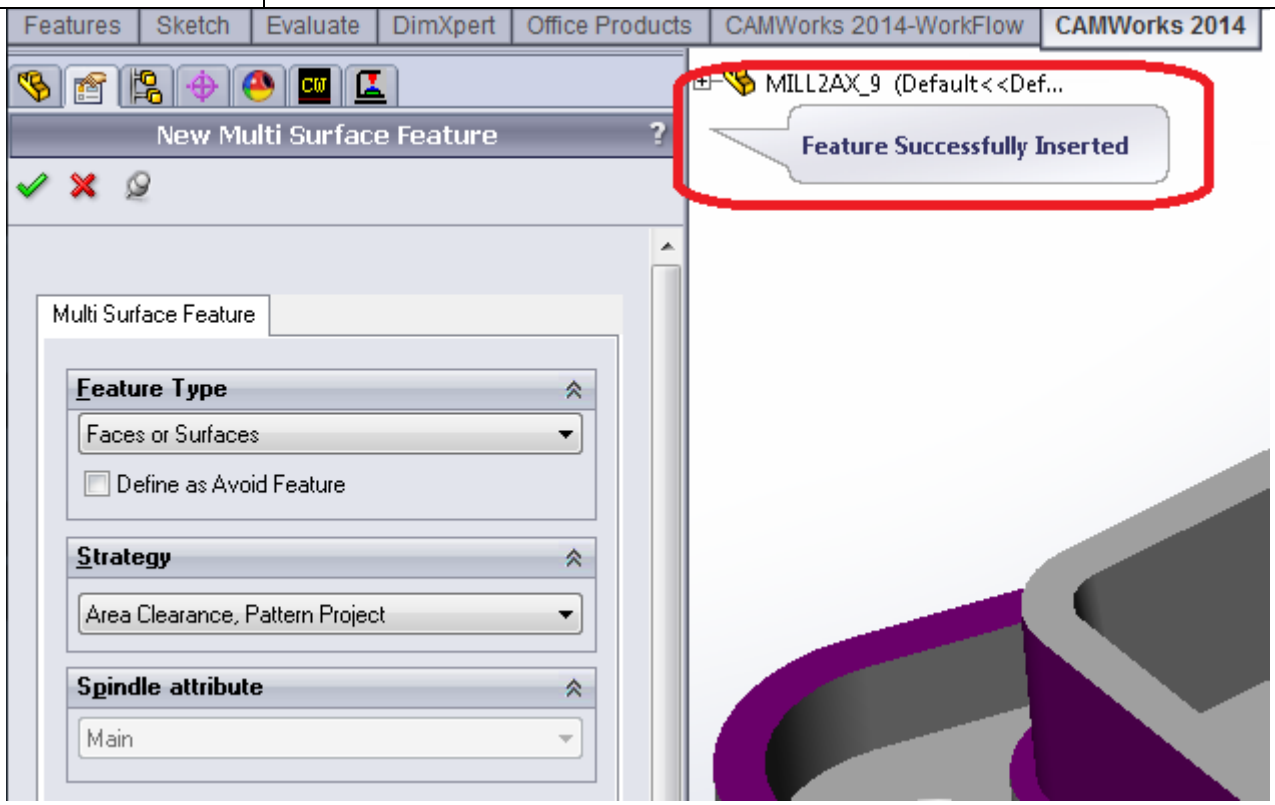
Purpose:	Provides options for viewing the 3D Model view of the Tool and Holder in the Preview Window when a tool is selected in the Tool Crib tab.
Implementation:	In CAMWorks 2014, a new functionality of displaying a Preview window containing the 3D Model view of the tool, whenever a tool/holder is selected in the Tool Crib tab of the Machine node dialog box, was introduced. This Preview window thus enables visual identification of the selected tool/holder. From CAMWorks 2015 version, this functionality has been enhanced to provide options for tool display. Additionally, options to view the corresponding Holder display are also provided. Users can now opt to change the display options for the tool and older to: • No display • Wireframe • Translucent • Shaded • Shaded with Edges



Preview Window

New - Message indicating successful insertion of interactively inserted Features

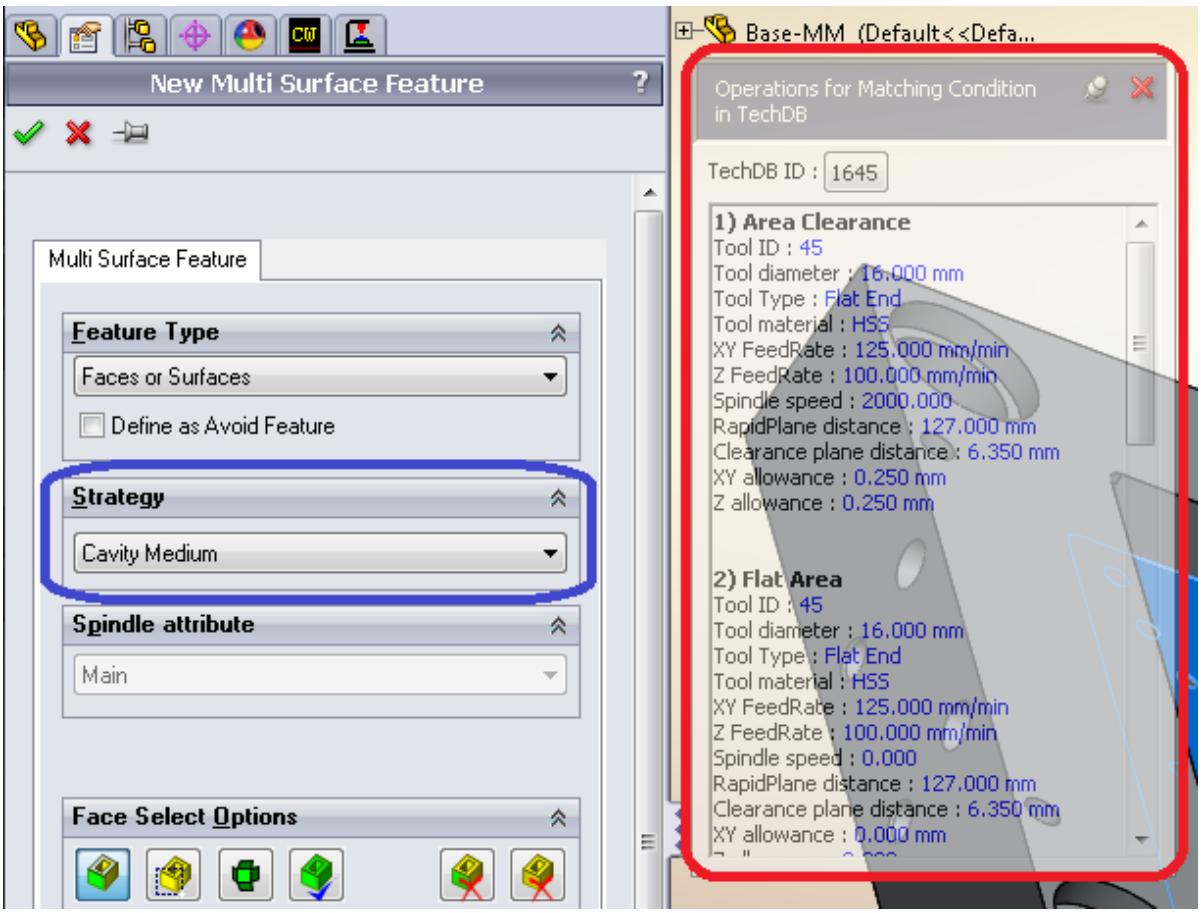
Purpose:	Provides an immediate feedback in the form of a bubble message when a Multi Surface Feature or Turn Feature is successfully inserted using Interactive Feature Recognition.
Implementation:	While using previous versions of CAMWorks, if the New Feature dialog box used for interactively inserting a Multi surface Feature or Turn Feature is in an unpinned state and user interactively inserts a feature, then the dialog box closes on successful insertion of the feature. The user can view the newly inserted feature in the CAMWorks Feature tree. However, if the New Feature dialog box is in a pinned state and the user clicks the Green check mark at the top left corner of the dialog box to insert the feature, then he/she has no way of knowing whether the feature got successfully inserted into the CAMWorks Feature tree or not. (Feature tree remain masked by the pinned New Feature dialog box). To overcome this limitation, from CAMWorks 2015 version onwards, a bubble message indicating successful insertion of the feature will be displayed when user clicks the green check mark at the top left corner of the pinned New Feature dialog box.



Bubble Message displayed on successful insertion of interactively inserted feature

New - Tooltip Window showing Operations for Selected Strategy

Purpose:	Provides an immediate feedback about the operations which will be generated for the selected feature based on the assigned strategy.
Implementation:	While working with parts in CAMWorks, the user assigns different strategies to the defined features. The user has no means of viewing the Machining sequence for the strategy. The user has to either browse through the TechDB or generate the operations and validate them with the requirement. To provide the user a way for viewing the machining sequence and associated Operation Parameters, a Tool tip window containing the details of the Machining Sequence will be displayed whenever the user assigns a strategy for a Mill, Turn or Wire EDM feature. **Appearance of the Tooltip Window** This Tool Tip window will be displayed when: - When user changes the strategy while defining or editing a Mill, Turn or Wire EDM Feature. - When user changes the assigned strategy in the *Feature parameters* dialog box of all features. - When user overs the mouse pointer over the *Strategy* field in the dialog boxes mentioned above. **Note:** If no operations have been defined for the defined feature in the TechDB, then the Tool tip window will display a message indicating that no operations will be generated for the feature since operations have not been defined for the particular feature condition in TechDB.

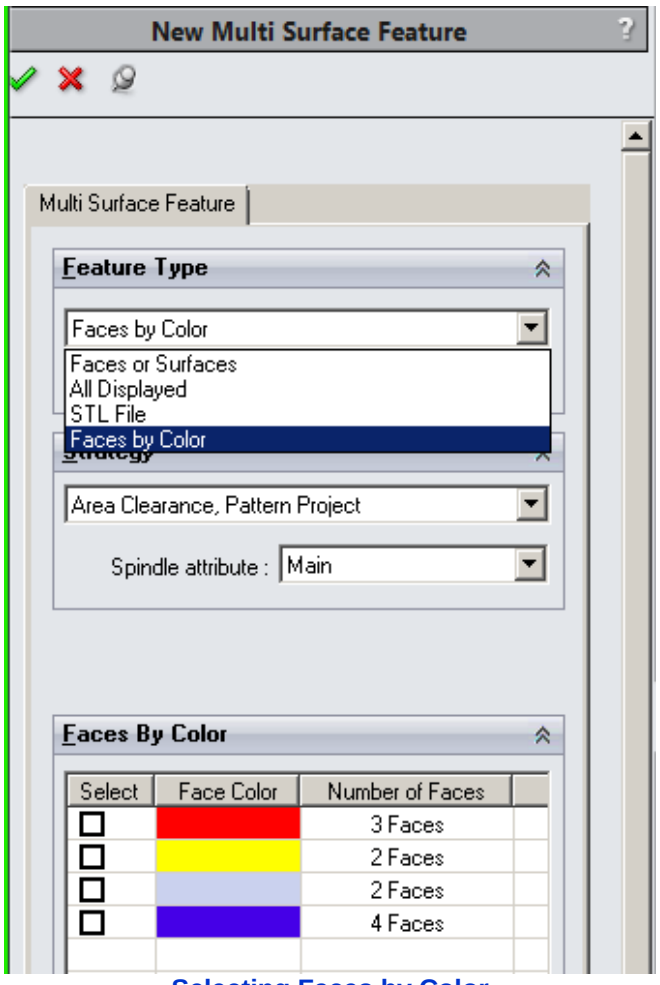


Tooltip Window displayed on assigning Strategy

Mill

New - Defining Multi Surface Features from Face colors of the part model

Purpose:	Provides the ability to define Multi Surface Features from surfaces with specific colors.
Implementation:	In previous versions of CAMWorks, multi-surface features could be interactively inserted by one of the following methods: • By selecting part faces individually in the graphics area • By selecting all the displayed faces • By using an STL file. From CAMWorks 2015 version onwards, multi surface features can be inserted by selecting faces of a particular color. This method is useful when you assign specific colors to the surfaces to be machined. When you select **Faces by color** as the Feature type in the *New Multi Surface Feature* dialog box, you can create a multi surface feature by selecting surfaces based on their colors. These colors and the number of faces with that particular color will be displayed in the *Faces by Color* group box. When the checkbox next to a particular color is checked, the multi surface feature will be created using faces with that color. You can select multiple colors, if required.



Selecting Faces by Color

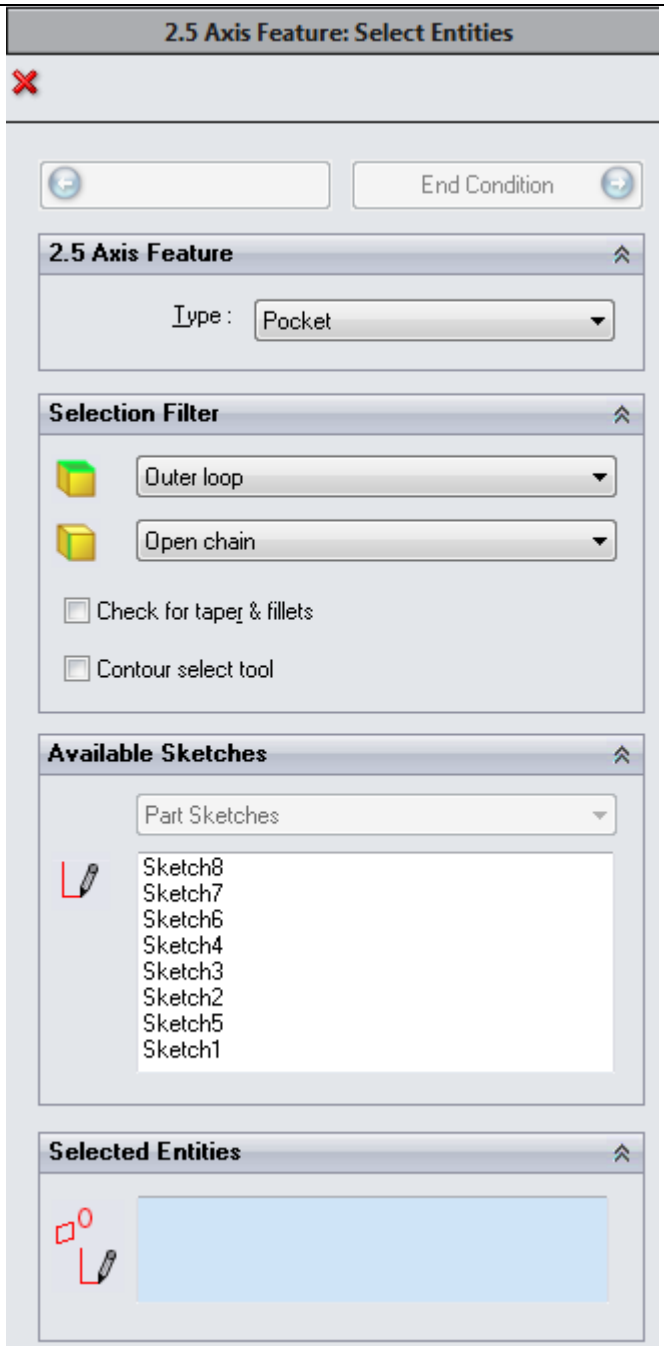
New - Provision to edit the base entity which defines profile of 2.5 Axis Mill Feature

Purpose:	Provides the ability to edit the base entity that defines the feature perimeter shape of an inserted feature.
Implementation:	In earlier versions of CAMWorks, once a 2.5 Axis Mill Feature was defined, parameters related to its end conditions, islands, etc. could be edited but there was no provision to edit the base entity that defined the feature perimeter shape. If the user wanted to modify the entity that defined the feature perimeter shape of a 2.5 Axis Mill Feature, then he/she had to delete that feature and create a new one. From CAMWorks 2015 onwards, user can edit the entities that define the feature perimeter shape of a 2.5 Axis Feature. To edit the feature, right-click on the feature item in the CAMWorks Feature tree and select the *Edit Definition* command from the context menu. The *Select Entities* dialog box of the 2.5 Axis Feature Wizard will be displayed. The entities that define the feature perimeter shape can be edited in the Selection Filter group box within this dialog box. You can also change the entity type used to define the feature. For example, if the feature perimeter shape was defined from a sketch, you can edit it so that feature perimeter is defined from a part model face or part model edges. **Note:** 1. You can edit the entities that define the feature perimeter shape but you cannot edit the feature type. The provision to edit the feature type will be made available in a future version of CAMWorks. 2. In the CAMWorks 2015 version, every parameter of a 2.5 Axis Mill feature except its feature type can be edited.

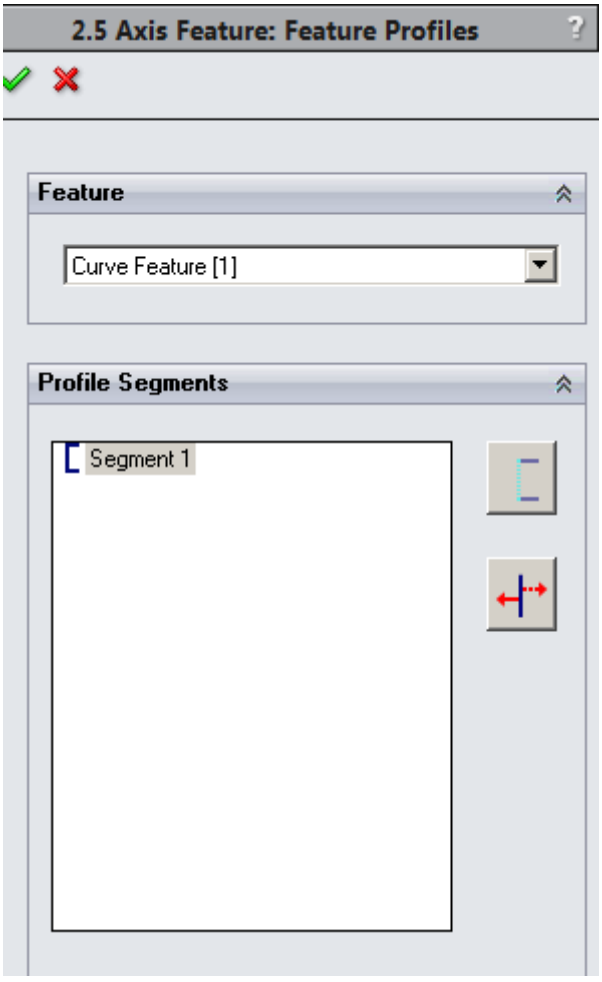
Improved - 2.5 Axis Feature Wizard UI updated to enable better graphics view

Purpose:	The UI of the 2.5 Axis Feature Wizard updated to suit the SOLIDWORKS UI.
Implementation:	In earlier versions of CAMWorks, the 2.5 Axis Feature Wizard (which is used to interactively insert/edit a 2.5 Axis Mill Feature) was available as a series of dialog boxes. The wizard was, by default, displayed in the graphics area and thereby occasionally obstructed the view of the part model in the graphics area unless the user manually shifted the wizard to another location on the screen. From CAMWorks 2015 onwards, the 2.5 Axis Feature Wizard will be available as a Property Manager Page on the left had side of the SOLIDWORKS/ CAMWorks Solids UI. This realignment ensures that the part model/assembly view in the graphics view is completely unobstructed. While the basic flow of the Wizard has remained unchanged, a few important changes have been implemented. These changes are listed below: • The dialog box **Feature & Cross Section Definition** has been renamed to **Select Entities**. • The dialog box **Edit Feature Profiles** has been renamed to **Feature Profiles**. • The **Island End Conditions** dialog box and **Select Island Entities** dialog box have been merged into one dialog box named **Island Entities** in order to facilitate smoother user interaction.

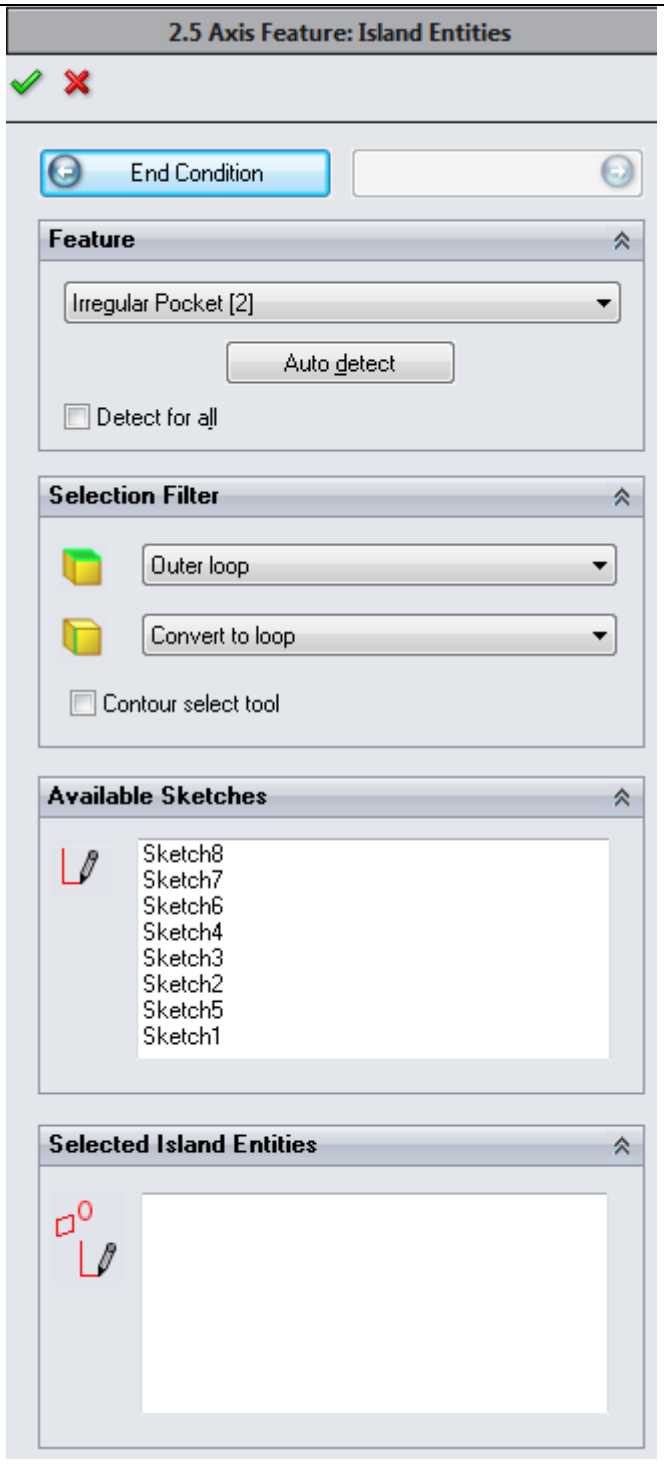
	• In previous versions, an option was provided in the 2.5 Axis Feature Wizard UI to create either single feature or multiple features (with single feature being the default option). From the 2015 version onwards, such an option won't be available in the UI. The default setting will allow you to insert multiple features of the same feature type. You can choose to insert one or more features depending on your requirements. • The **Check for tapers and fillets** option will be enabled only when inserting a single feature interactively. If multiple feature profiles are defined in the *Feature Profiles* dialog box in order to insert multiple features, then the *Check for tapers and fillets* option will be disabled.



Select Entities Dialog box



Feature Profiles Dialog box



Island Entities Dialog box

Note:
The functionality of inserting multiple features interactively is disabled in the following cases:

1. When the **Feature type** is set to Open Pocket or Face Feature.
2. When defining a tapered feature (i.e. when the *Check for tapers and fillets* checkbox option is checked in the UI).

New - Assigning Multiple Sets of Cutting Parameters to Tools

Purpose:

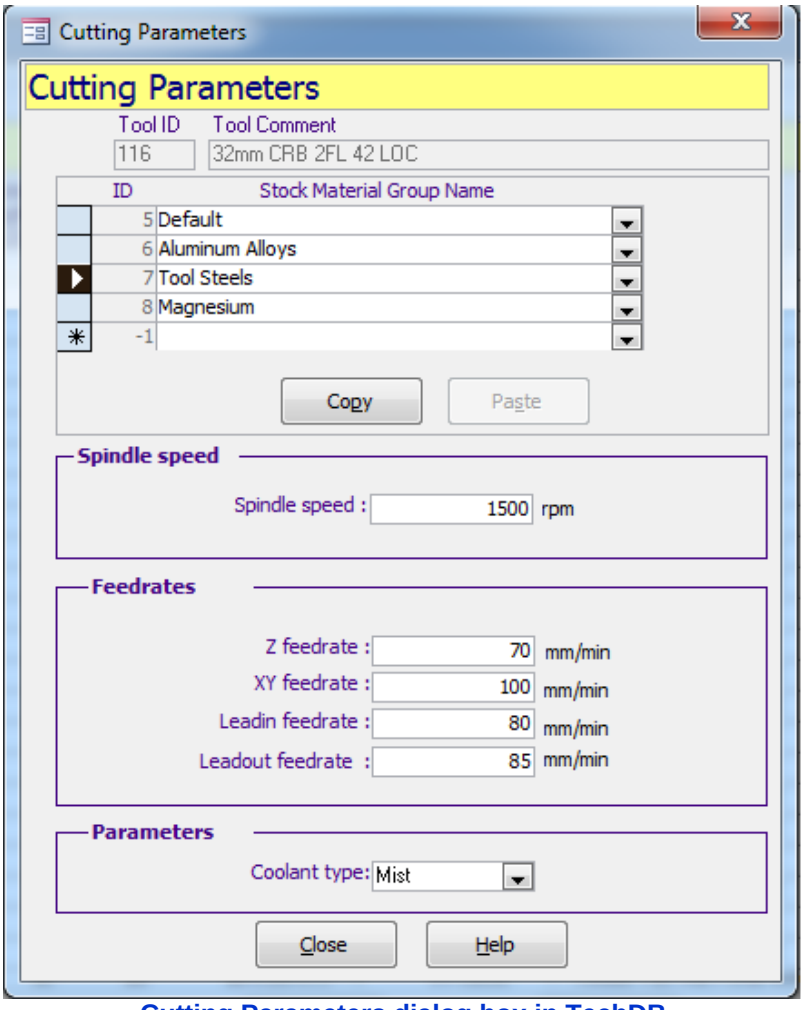
Provides the ability to define several sets of cutting for each Mill tool and point-to-point tool based on the selected Stock Material type.

Implementation:

In previous versions of CAMWorks, only one set of Cutting Parameters could be assigned to each Mill tool and point-to-point tool in the TechDB. From CAMWorks 2015 onwards, several sets of cutting parameters can be assigned for each tool within the TechDB. Each set will be labelled and distinguished based on the Stock Material Type.

Defining multiple sets of cutting parameters for Mill and pint-to-point tools in the TechDB:

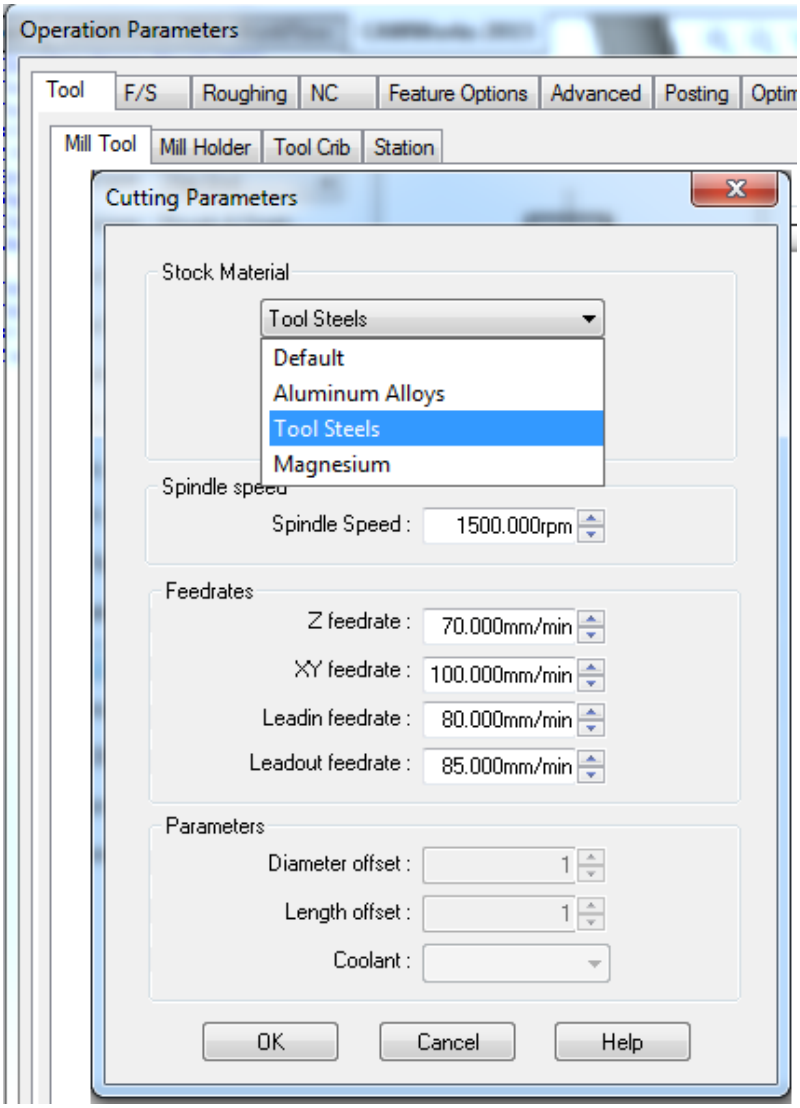
The cutting parameters for each tool are defined in the TechDB. The Cutting Parameters dialog box is displayed when you click the Cutting parameters in the Tool form when defining Mill and point-to-point tools.
You can assign multiple Stock Material groups to the tool in this dialog box. For each stock material group, you can view/modify/save the cutting parameters.



Cutting Parameters dialog box in TechDB

Selecting desired set of Cutting Parameters in the CAMWorks UI:
Once the tool has been assigned to an operation, the Cutting Parameters can be viewed and modified in the Cutting Parameters dialog box. (This dialog box is displayed by executing the Cutting Parameters command of the Mill Tool tab in Operation Parameters dialog box).

- Depending on the Stock material type selected in the Stock Material dropdown list, the corresponding cutting parameters from the TechDB will be loaded in the dialog box.
- The displayed cutting parameters can be modified for the operation. However, these edited values will be applied to the current operation only and will not be saved back to TechDB.
- If the *Associate with stock material option* is enabled, then the stock material will be updated every time the stock material is updated in the Stock manager dialog box.



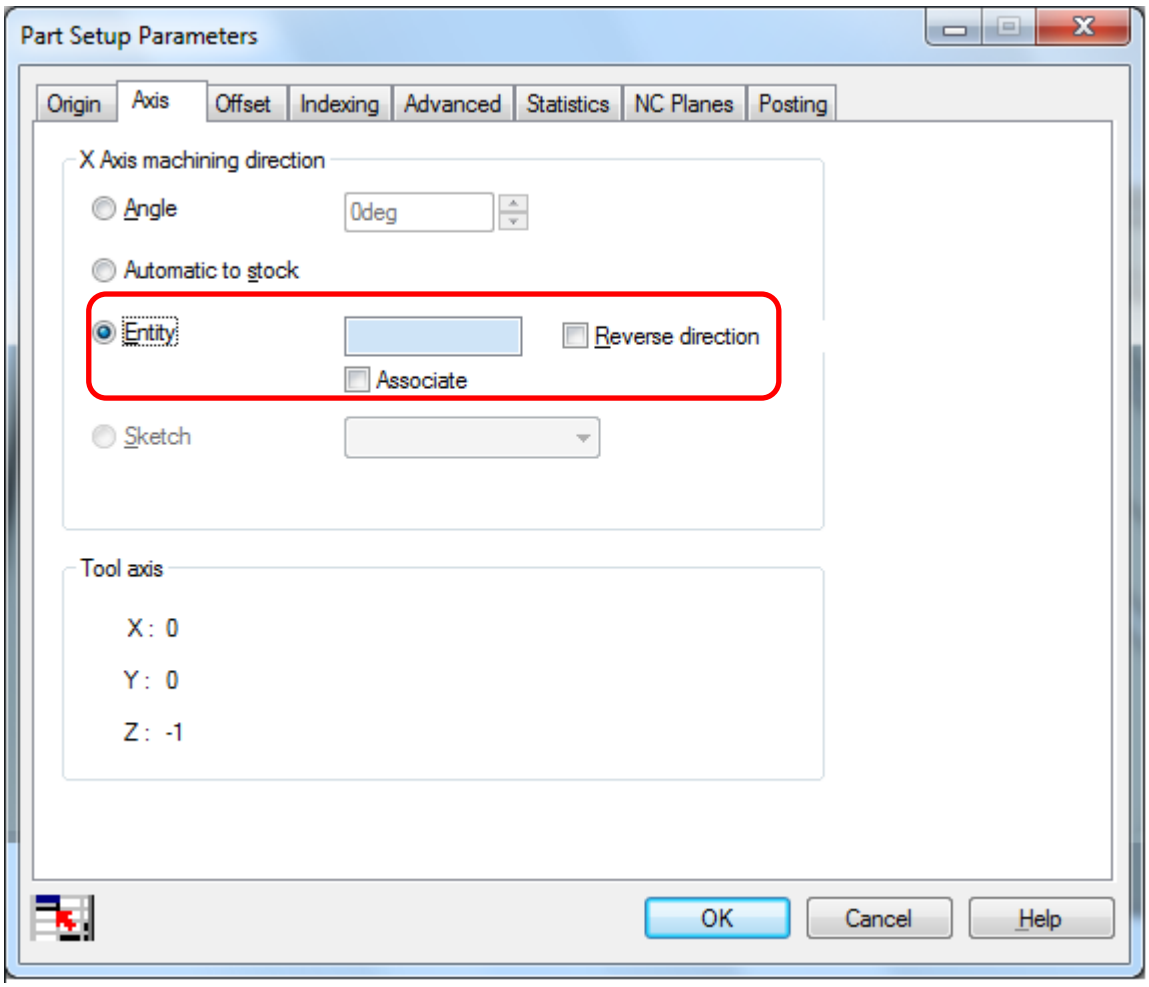
Cutting Parameters dialog box in CAMWorks UI

Improved - 2.5 Axis Limited Module now supports Setups in multiple directions

Purpose:	Enhances the capability of the 2.5 Axis Limited License module by supporting Mill Part Setups in multiple directions
Implementation:	In previous versions of CAMWorks, users who purchased only the 2.5 axis Limited module of CAMWorks (and not other advanced modules) faced the following limitations: 1. Only one Mill part Setup is detected as per the defined Coordinate system or the SOLIDWORKS Coordinate system. 2. User had to create multiple instances of the part in order to machine the part with multiple setups. From the CAMWorks 2015 version onwards, these limitations are addressed. On executing the Extract Machinable Features command, all possible Mill Part Setups and features will be detected. User will be able to generate toolpaths and operations for all the features.

New - Defining X Axis of Mill Part Setup by picking an entity

Purpose:	Provides the ability to define the X Axis of a Mill Part setup by selecting a graphical entity.
Implementation:	The direction of the X axis of a Mill Part Setup is defined in the Axis tab of its corresponding Part Setup Parameters dialog box. In earlier versions of CAMWorks, the direction of the X Axis could be defined from an angle relative to SOLIDWORKS, longest edge of the stock, linear edge of the part model or sketch. From CAMWorks 2015 onwards, the option to define the X Axis direction from a model edge has been replaced with the more inclusive option of defining the X Axis direction from a graphical entity. The **Edge** option has been replaced with **Entity** option. The selected graphical entity can be a Reference Plane, Reference axis, linear model edge, planar face, cylindrical face, linear sketch segment or Co-ordinate system. The ID of the selected entity will be displayed in the field next to the Entity option. **Note:** 1. If the Associate option is checked, then the selected entity will be associated with the corresponding part entity. 2. The direction of the X axis can be reversed using Reverse Direction option. 3. If a cylindrical face is selected as the entity, then the axis of the cylindrical face will be assigned as the X direction. 4. Circular edges cannot be used to define the direction of the X axis. If any such unsuitable entity is selected for defining the X axis, then the error message "*Wrong Entity has been selected to define X direction, please select another entity*." will be displayed.



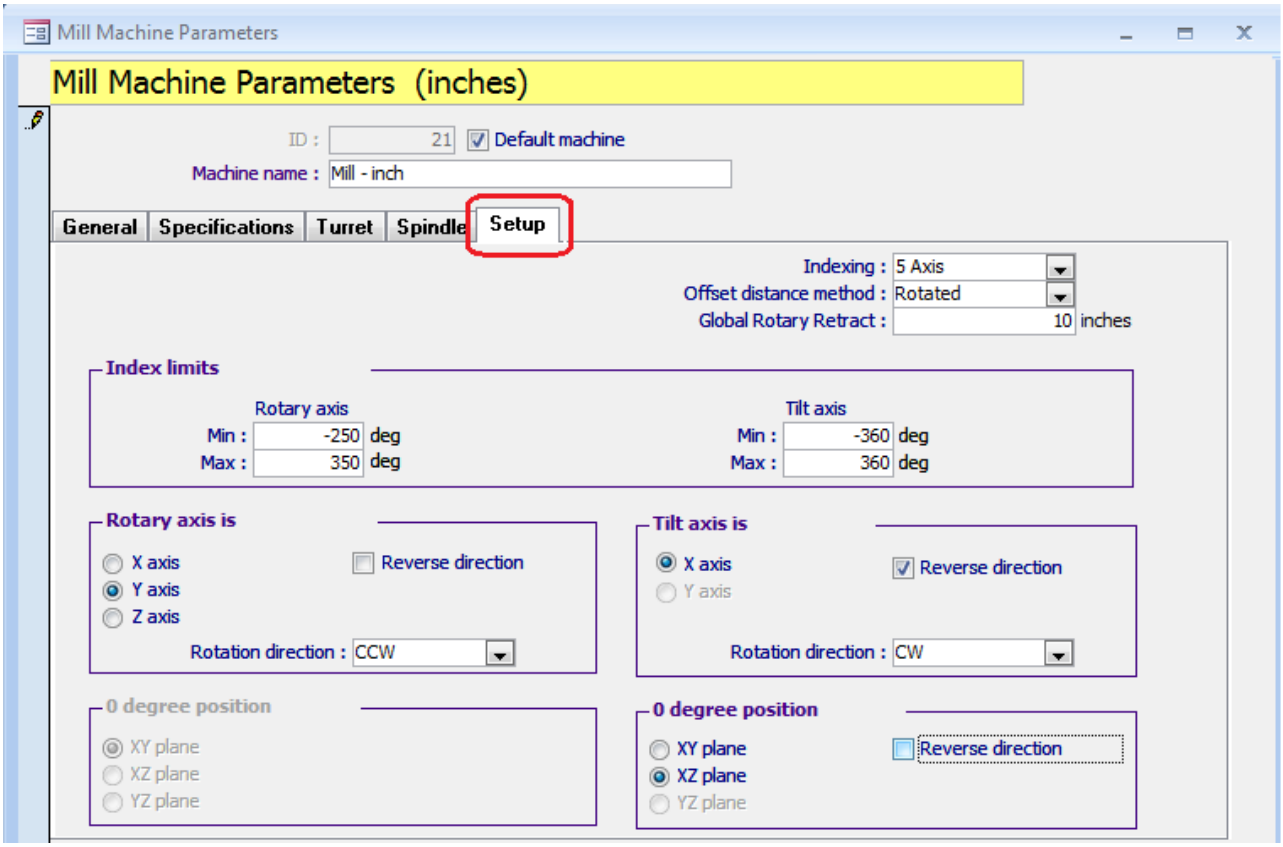
Entity option in the Axis tab of Part Setup Parameters dialog box

New - Assigning Rotary & Tilt Axis parameters to Mill Machine Definition in TechDB

Purpose:	Functionality to assign Rotary Axis and Tilt Axis parametric values in Mill Machine Definition Form of **TechDB**.
Implementation:	In earlier versions of CAMWorks, within the TechDB, there is no provision to define Rotary axis and Tilt axis parameters for the Mill Machine definition. Consequently, every time the user wants to machine a part/assembly on a 4 Axis Mill machine or 5 Axis Mill Machine, he/she has to assign values to the Rotary Axis and Tilt Axis parameters in the Machine Definition dialog of the CAMWorks UI. At facilities where 4 Axis/5 Axis Mill Machines are used, assigning rotary and tilt axis parameters every time a part/assembly is machined proves a needlessly repetitive task. Mill machines are standard and their configurations don't change frequently. A provision to assign default values to the Rotary and Tilt axis parameters in the Mill Machine definition within TechDB will eliminate the need to define rotary and tile axis parameters every time the part/assembly is machined. Such a provision has been made in the TechDB from the CAMWorks 215 version onwards. In the Mill Machine Parameters form within TechDB, a new tab "Setup" has been added. Default values for Rotary and Tile Axis parameters can be assigned in this tab.

Note:

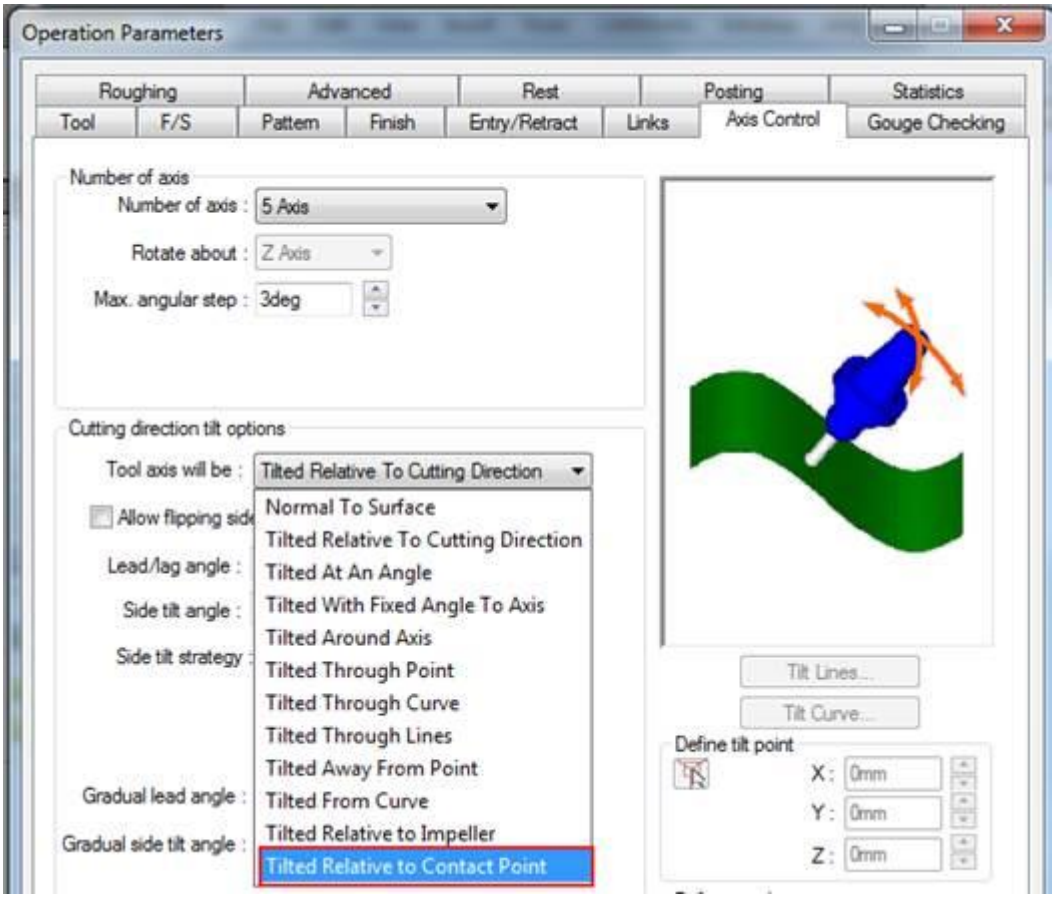
1. In the Setup tab in the Mill Machine Parameters form, Rotary and Tilt axis parameters will be visible only if the Indexing is set to **4 Axis** or **5 Axis**. They won't be visible when Indexing is set to **None**.
2. When machining a part/assembly, if you select a 4 Axis or 5 Axis Mill Machine, the Rotary and Tilt Axis parametric values assigned in the Setup tab of the corresponding Mill Machine Parameters form in the TechDB will be loaded as default values in the Rotary Axis tab and Tilt axis tab of the Machine dialog. If required, these default values can be overridden by reassigning the values in the Rotary Axis and Tilt Axis tab.



Setup tab of Mill Machine Parameters Form in TechDB

New - New Tool Tilting option for Cutting Direction of Multiaxis Mill Operations

Purpose:	New Tool Tilting option for Cutting direction of Multiaxis Mill operations
Implementation:	A new Cutting Direction tilt option "Tilted Relative to Contact Point" has been made available in Axis Control tab of Operation Parameters dialog box for Multiaxis Mill operations. When this option is selected, you can also specify the Lead Angle and Lag angle.



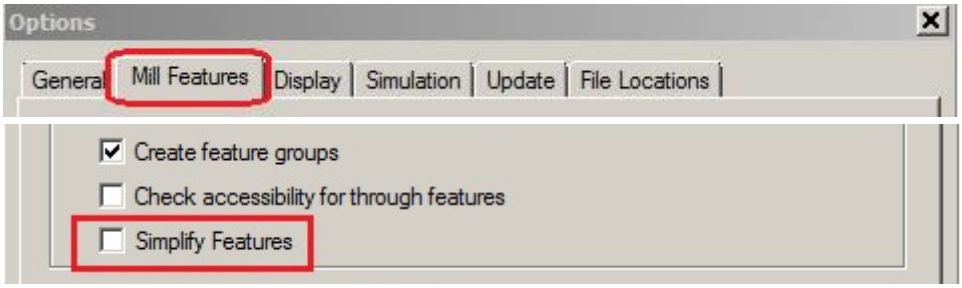
New Tool Tilting option for Cutting Direction: 'Tool Relative to Contact Point'

New - Option to identify intersecting Hole and Pocket/Slot features separately

Purpose:	Provides an option to identify circular sections that intersect with the perimeter of a pocket or slot feature as hole features which are otherwise clubbed with the pocket/slot and identified as irregular shaped features.
Implementation:	For solid models that have one or more hole features intersecting with a pocket/slot feature, when the Extract Machinable Feature command is run, the resultant feature recognized by AFR will be an irregular pocket/slot feature. In certain cases, (as in the case of the Die & Mould industry), users may want AFR to separately identify these intersecting features as Hole features and Pocket/Slot features. Up to CAMWorks 2014 version, users with such a requirement had to use SOLIDWORKS sketches in order to define the desired features and use hole features to machine the corners.

From CAMWorks 2015 version onwards, the Features tab in the CAMWorks Options dialog box provides the **Simplify Features** option for this purpose.

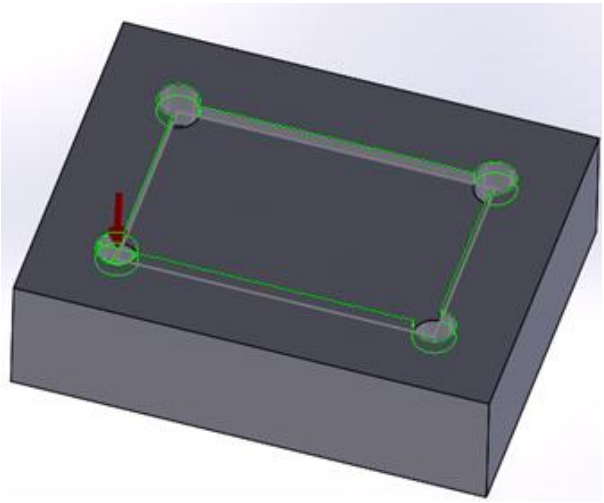
When this option is enabled, AFR will identify all circular sections intersecting with the perimeter of the pocket or slot feature as Hole features.



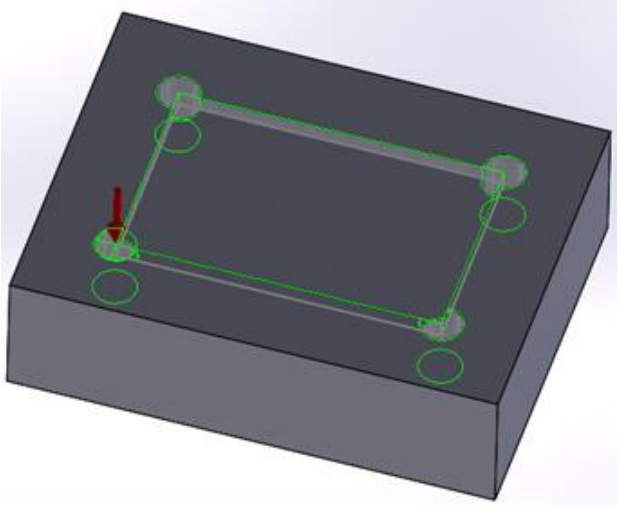
Note:

The 'Simplify Features' option will be active in the Features tab of the CAMWorks Options dialog box only when the following conditions are fulfilled:

- The Machine type selected is a Mill Machine.
- The Feature type of 'Non Holes' is selected in Features tab in the CAMWorks Options dialog box.



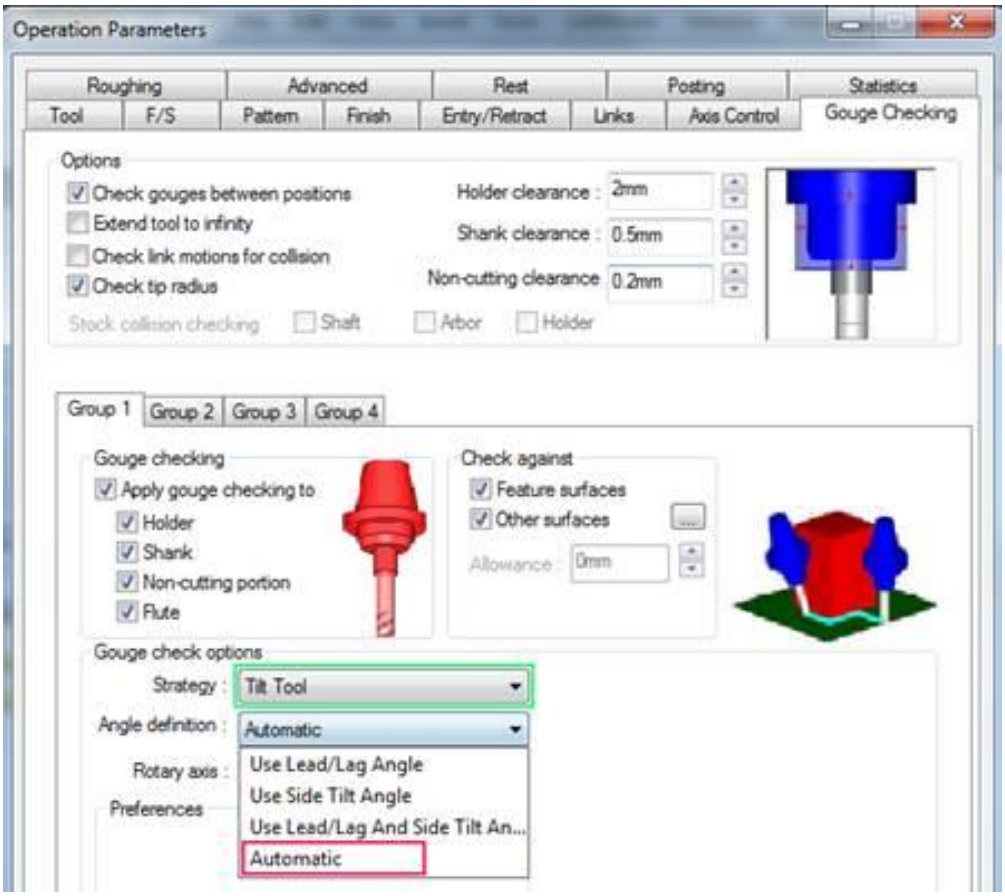
Irregular pocket Feature identified when 'Simplify Features' option disabled



Rectangular Pocket and Hole Features identified when 'Simplify Features' option enabled

Improved - Gouge Checking Strategy for Multiaxis Mill Operations

Purpose:	Enhance "Angle Definition" options for the Gouge Checking Strategy of "Tilt Tool" by making an additional option available for Ball Nose Tools.
Implementation:	For Multiaxis operations, applying a gouge checking strategy prevents gouging by tilting the tool away from the surface. From CAMWorks 2015 version onwards, the Gouge Checking strategy "**Tilt Tool Away With Max Angle**" has been renamed to "**Tilt Tool**". The 'Tilt Tool strategy' provides gouge checking by assigning Angle Definition parameters. **"Automatic" option for Angle Definition of 'Tilt Tool' strategy** From CAMWorks 2015 version onwards, for 'Tilt Tool' gouge checking strategy, an additional option "Automatic" has been provided for Angle Definition. This option is available only for Ball Nose Mill tools. When this option is selected, it enables tilting of the tool to avoid gouging the feature. Under circumstances when a gouge cannot be avoided, the portion of the toolpath resulting in the gouge will be deleted.



New Angle Definition option 'Automatic' for 'Tilt Tool' Gouge Checking Strategy

Turn

New - Tool Crib Priority Option for Turn and Mill-Turn

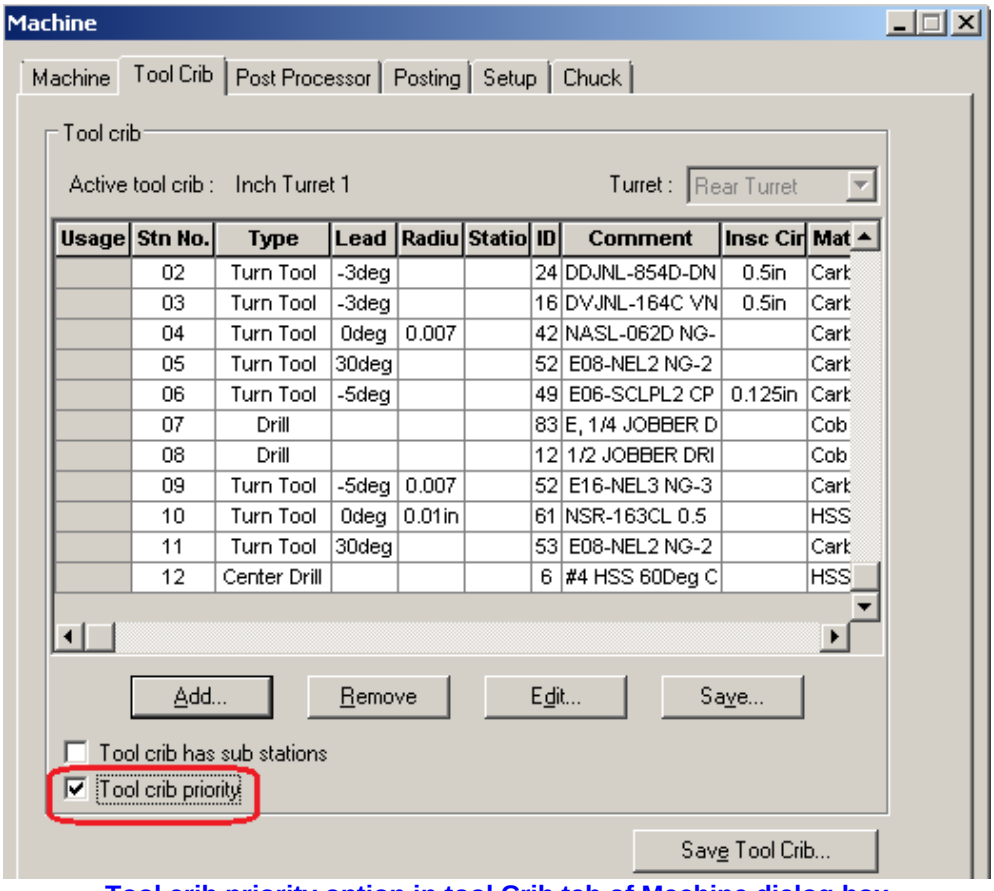
Purpose:
When assigning tools to operations, provides an option to modify the tool selection rules so that a higher priority on assigning tools from the tool crib.

Implementation:

CAMWorks is the ability to automatically assign tools to machining operations for each individual feature type. These rules are user-defined and are specified in the Technology Database under the Feature & Operation menu.
When the Tool crib priority option in the Tool Crib tab of the Machine Dialog Box is selected, the automatic tool selection rules are modified to place a higher priority on selecting tools from the tool crib. CAMWorks will consider all tool selection criteria (TechDB ID, Tool Type and Holder Orientation) and major tool parameters while selecting the tool from the Tool Crib. If an exact match isn't found, then certain rules of the tool selection process (such as TechDB ID, Holder Type, Nose Radius, Include Angle) are relaxed. If a matching tool isn't found in the tool crib even after relaxing those rules, then CAMWorks will add the tool from the tool library.

Note:

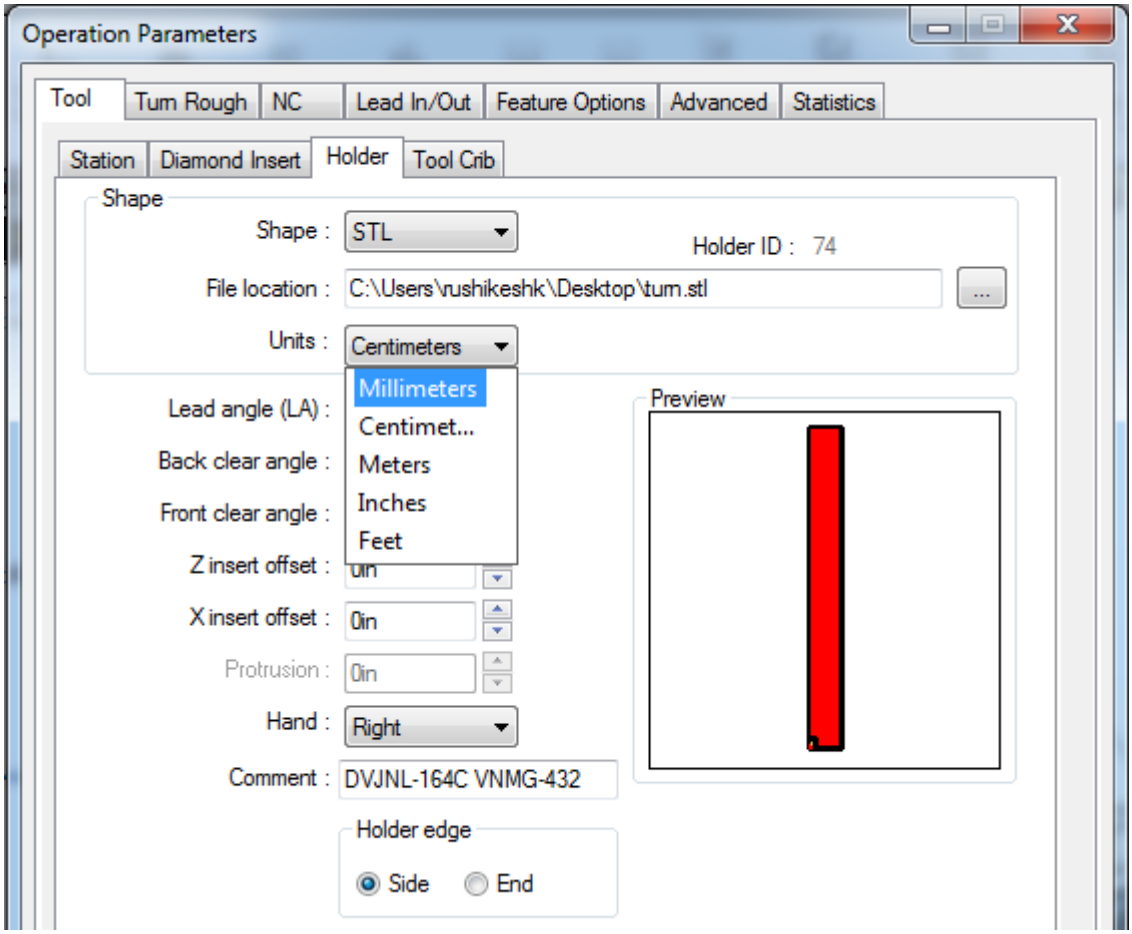
- During the Tool Selection Process with Tool Crib Priority option enabled, if multiple tools satisfying the Tool Selection criteria are available, then the first tool in the tool crib matching the selection criteria will be assigned to the operation.
- If the Tool crib priority option is enabled, then in the CAMWorks Operation tree, a red mark will appear over the icon of the Operation for which a tool has been assigned from the Tool library instead of the Tool Crib. This mark serves as a visual indicator for tools assigned from the Tool library.



Tool crib priority option in tool Crib tab of Machine dialog box

Improved - Assigning Units to STL Files used for defining Turn Holder/Boring bars

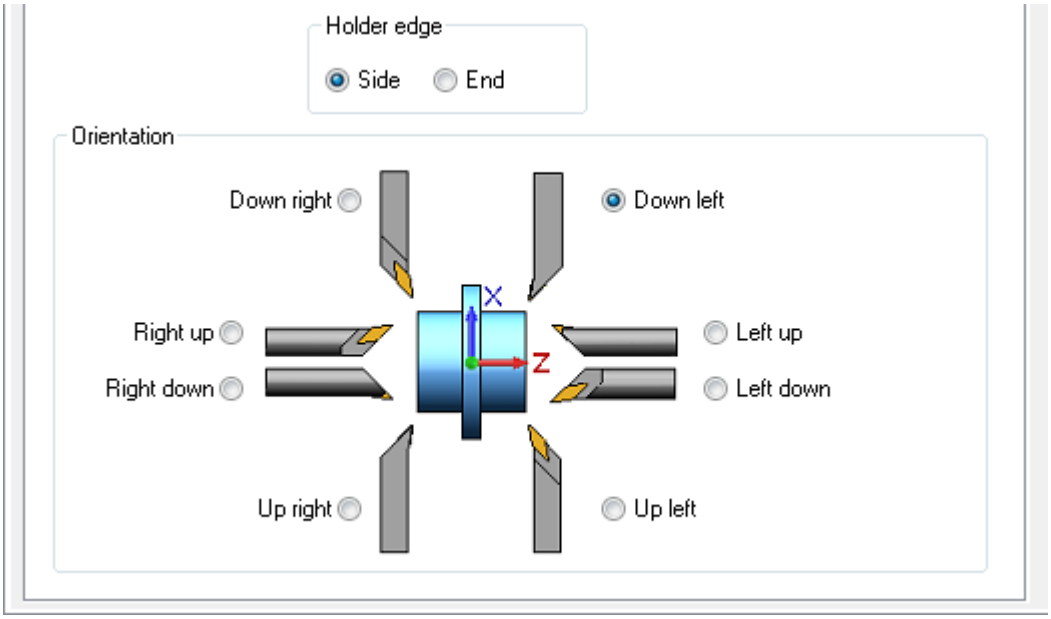
Purpose:	For Turn Holder/ Boring Bar defined from STL files, provides the ability to change the units of STL files.
Implementation:	One of the methods provided by CAMWorks for defining the geometry of a user-defined Turn Tool Holder or Boring Bar is by selecting an STL file. These STL files do not have any units. In previous versions of CAMWorks, it is assumed that that the STL file units are in meters. Often, users receive these files from the manufacturer and have no control on the units. From CAMWorks 2015, functionality to assign units is supported for STL files. This functionality allows the imported STL model to be transferred with the required scale model factor.



Changing Units of STL File

Improved - Controls for editing Holder Edge and Holder orientation

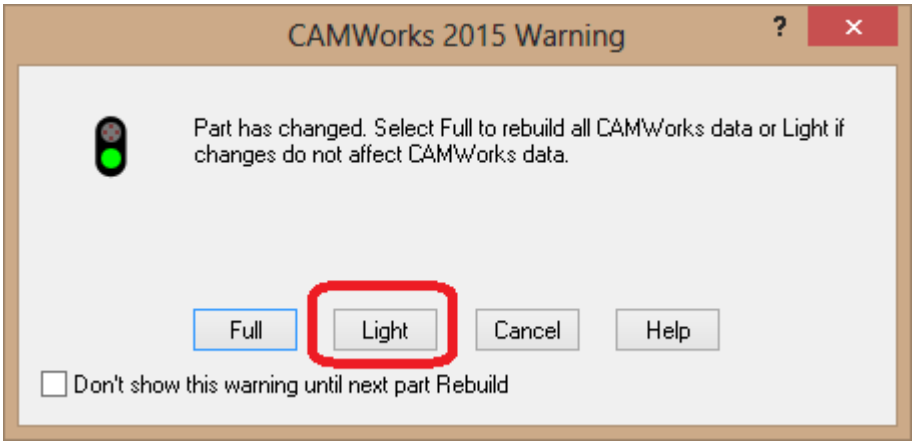
Purpose:	Enables the controls for editing the Holder Edge and of Cut and Holder Orientation parameters when editing a Turn Insert in the Tool Crib tab of the Machine dialog box.
Implementation:	In the Tool Crib tab of the Machine dialog box, when user highlights a tool in the Tool crib and clicks the Edit button, the Edit Tool Parameters dialog box is displayed. This dialog box allows the user to view/edit the Tool and Holders parameters for the current part being machined. In previous versions of CAMWorks, within the Holder tab of the Edit Tool Parameters dialog box, the 'Holder Edge' parameter wasn't available and the Holder Orientation parameter was disabled. Consequently, users couldn't view the default tool orientations as defined in the TechDB. The only way to view and edit these parameters was by applying the tool to an operation and then viewing/editing them in the Holder page of the Tool Crib tab of the Operation Parameters dialog box. From CAMWorks 2015 onwards, these parameters can be viewed and edited in the Holder tab of the Edit Tool Parameters dialog box.



Holder Edge and Holder Orientation Parameters

New - Light Rebuild Option in Turn Module

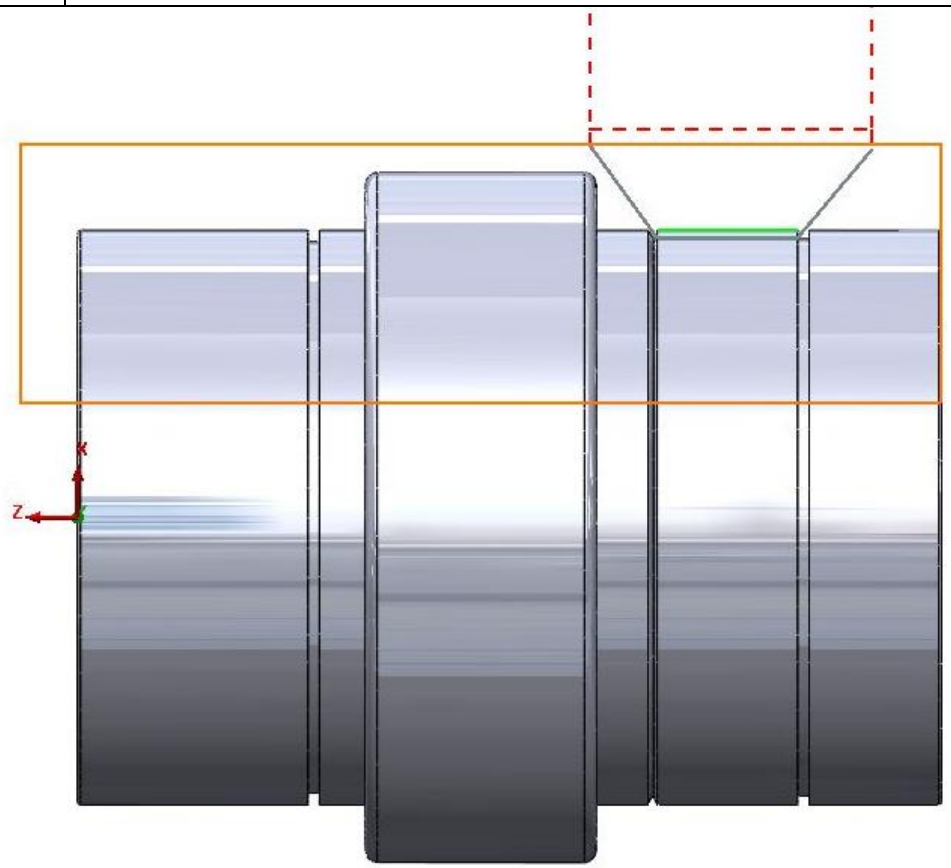
Purpose:	Provides the ability to Light Rebuild CAM data in Turn module
Implementation:	The Rebuild functionality in CAMWorks provides the ability to automatically update Machinable features, operations and toolpaths after the size, shape or sketches of a part model have been modified. In earlier versions of CAMWorks, for Mill module and Wire EDM module, two Rebuild functionalities are provided viz. the Full Rebuild and Light Rebuild. However, for the Turn module, only the Full Rebuild functionality is provided. • A Full Rebuild updates the following: a. Stock size and shape b. Setup Origins c. Automatically recognized features d. Interactively inserted features e. Operations and toolpaths associated with all the features. • A Light Rebuild performs all the actions of Full Rebuild except automatically Recognized Features. From the CAMWorks 2015 version, the Light Rebuild functionality has been extended to the Turn Module too.



Warning Message shown on Rebuild

New - Support Lead In/ Out Parameters for Turn Thread Operations

Purpose:	Makes the Lead in and Lead Out options available for Turn Thread toolpaths as they will aid in generating better quality threads.
Implementation:	A new Lead In/Out tab is available in the Operation Parameters dialog box of the Thread operation. You can set the desired Lead In/Out values using the parameters in this tab. **Note:** • When you set the option of "Canned cycle output" to true, then all the options in the Lead In/Out tab will be disabled. • Within the TechDB, the default Lead In/Out parameters for Turn Thread operations can be defined in the Lead In/Out tab of the Turn Thread Operation Parameters form.



Turn Thread Toolpath with Leadin and Leadout

New - Option to control Chuck Location Definition for Turn Setup

Purpose:
Provides an option to control whether Chuck location is to be defined or not.

Implementation:

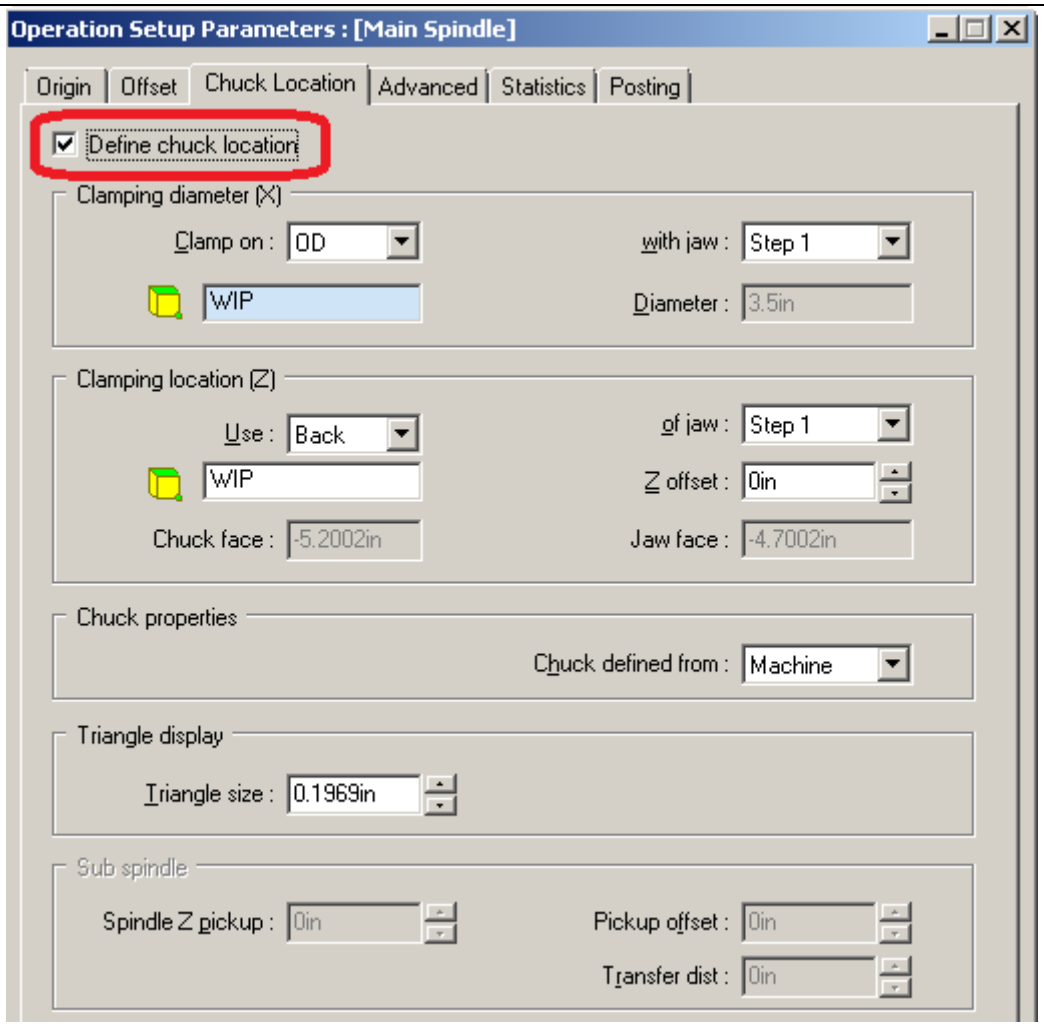
In the *Chuck Location* tab of the *Setup Parameters* dialog box, the **Define Chuck location checkbox** has been provided.

- When this checkbox is unchecked, all the parameters in the Chuck location tab will be disabled. As a result, chuck location cannot be defined.
- When this checkbox is checked, all the parameters in the Chuck location tab will be enabled. Consequently, chuck location can be defined using the parameters in this tab.

Note:

1. For any new turn part being machined, the *Define chuck location* checkbox will be unchecked by default.
2. When any legacy Turn or Mill-Turn part file (for which chuck location has been defined) is opened, the *Define chuck location* checkbox will be checked.

When to uncheck the 'Define chuck location' option:
If Chuck Location information is defined for a Turn/Mill-Turn part, then it will be accessed and utilized by Sub Spindle Operations used for machining the part. If certain cases, you might not want the Sub Spindle Operations to use the Chuck Location information. To prevent the Sub Spindle operations from using the Chuck Location Information, uncheck the *Define Chuck Location* option.



Define Chuck Location checkbox in Chuck Location tab of Operation Setup Parameters dialog box

New - Provision to assign default values to Offset Parameters for Bar Stock

Purpose:
In Turn and Mill-Turn module, provides a functionality in the Stock Manager UI wherein user can assign default values to the Offset Parameters when stock type is set to Bar stock (cylindrical bar).

Implementation:

In Turn and Mill-Turn module, when stock type is defined as a bar stock, user can assign offset values to the Bar Stock in the Stock Manager UI. The Offset parameters to which these values can be assigned are Diameter
Offset, Front Face of Stock Offset and Back Face of Stock Offset.

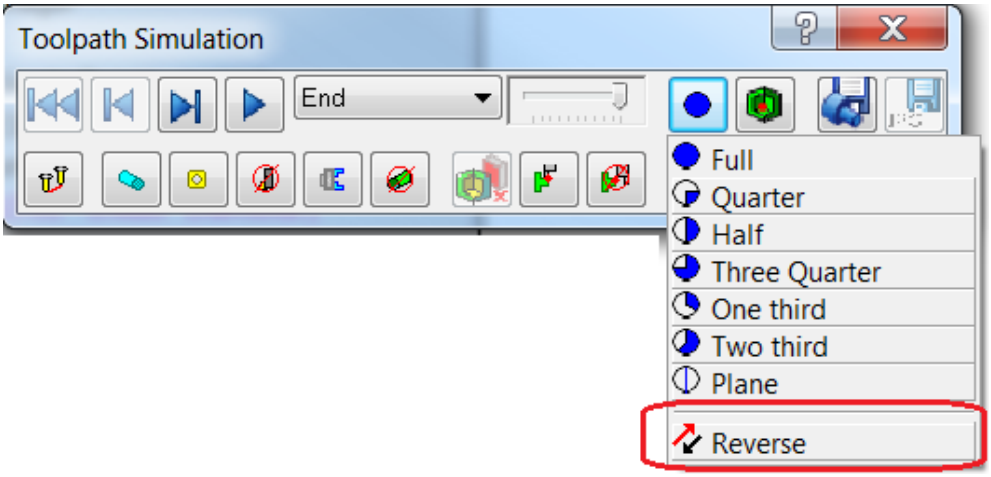
Two new buttons viz. **Set as Default** and **Load Default** have been provided in the Stock Manager UI.

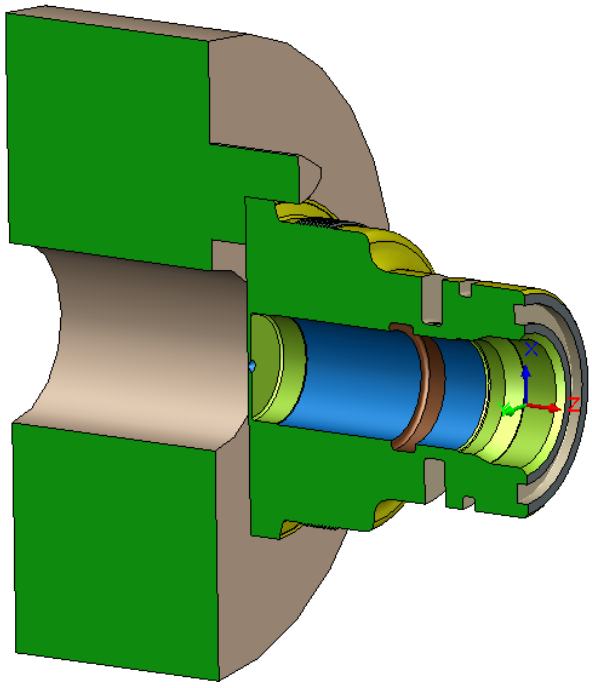
- When CAMWorks is launched for the first time as an Add-In within SOLIDWORKS/CAMWorks Solids, the default values for all Offset Parameters in the Stock Manager UI will be set as zero. These values will be retained until you assign another set of values as the default values using the **Set as Default** command button.
- When you click the **Set as Default** button, the current Offset Parameter values assigned will be saved as default values. The default Stock Offset Parameter values thus saved can be assigned to other Turn or Mill-Turn stocks by clicking on the *Load default* button.
- When you click the **Load Default** button, the default Offset Parameter values assigned will be loaded into the respective fields. (If Offset parameter values are already assigned in the Offset Parameters fields, then those values will be replaced with the default values when the *Load default* button is clicked.)

Turn Stock Manager Dialog Box

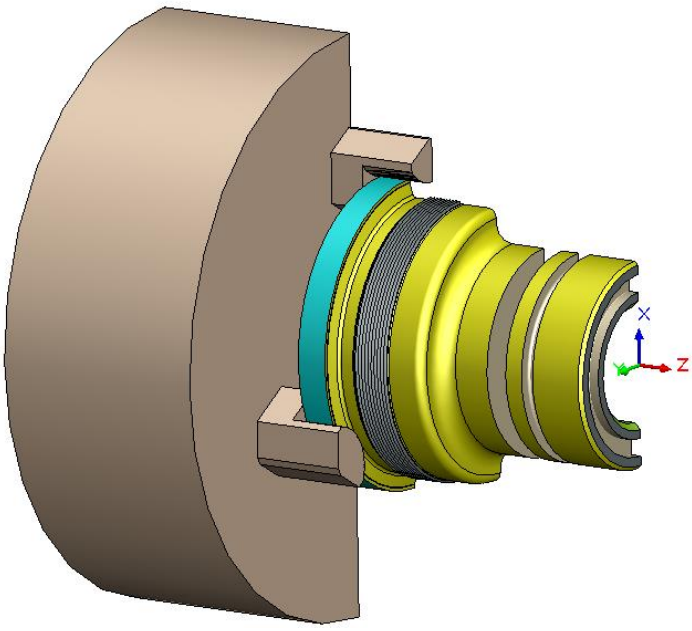
New - Reverse command in Turn Simulation mode for reversing the sectioning direction

Purpose:	In Turn/ Mill-Turn Simulation mode, provides a Reverse option in the Section View menu. This command allows the user to reverse the direction of sectioning.
Implementation:	During Section View simulation of Turn and Mill-Turn parts, a Reverse option is provided in the Simulation Toolbar's Section View menu. Executing this Reverse command enables the user to reverse the direction of sectioning. This option is especially useful in cases where the sectioned portion of the part is away from the operator's view. It eliminates the need to rotate the part for viewing the sectioned portion.





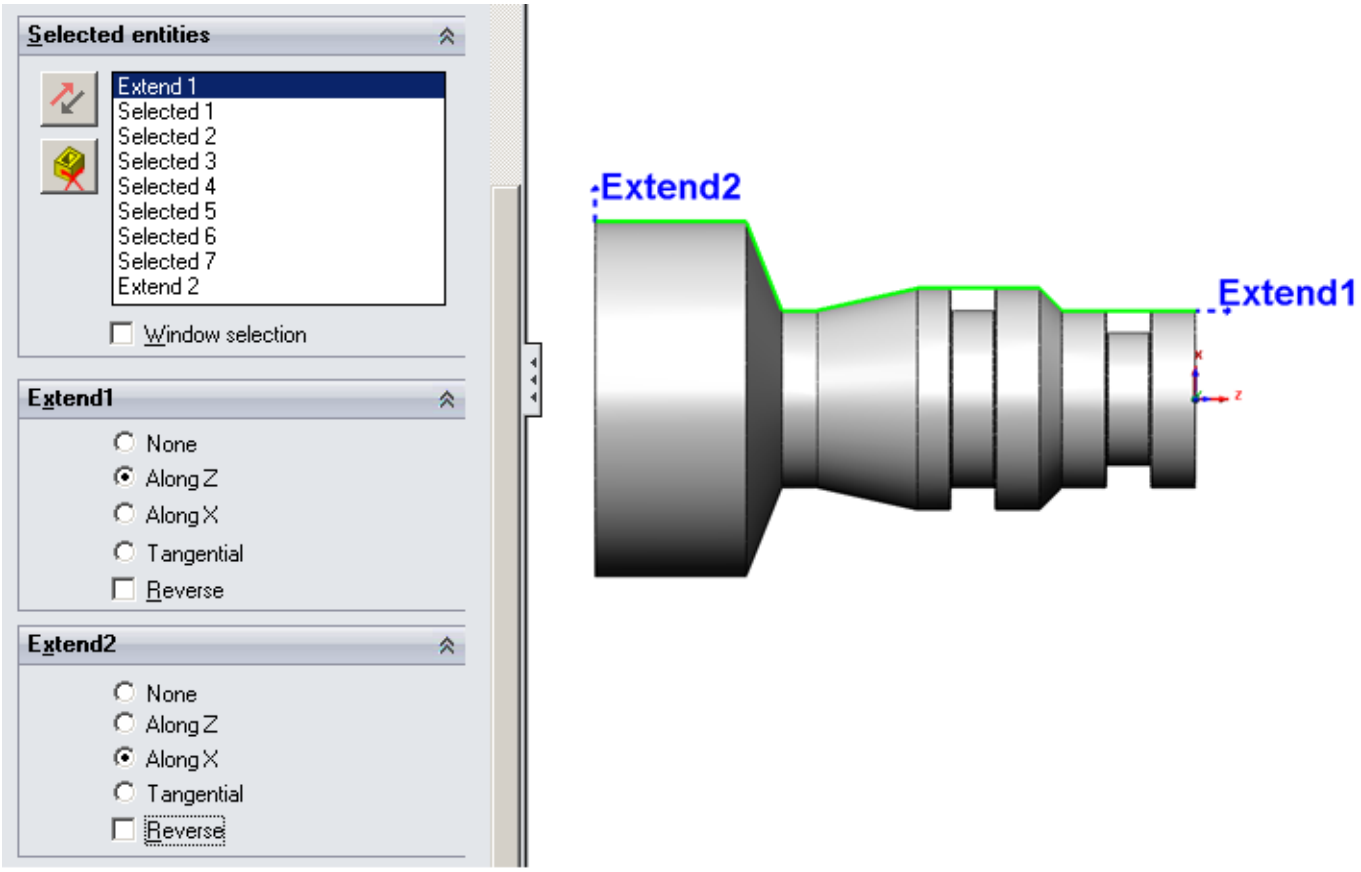
Half Section View



Half Section View after Reversing

Improved - Labelling of Turn Feature Extensions displayed in graphics area

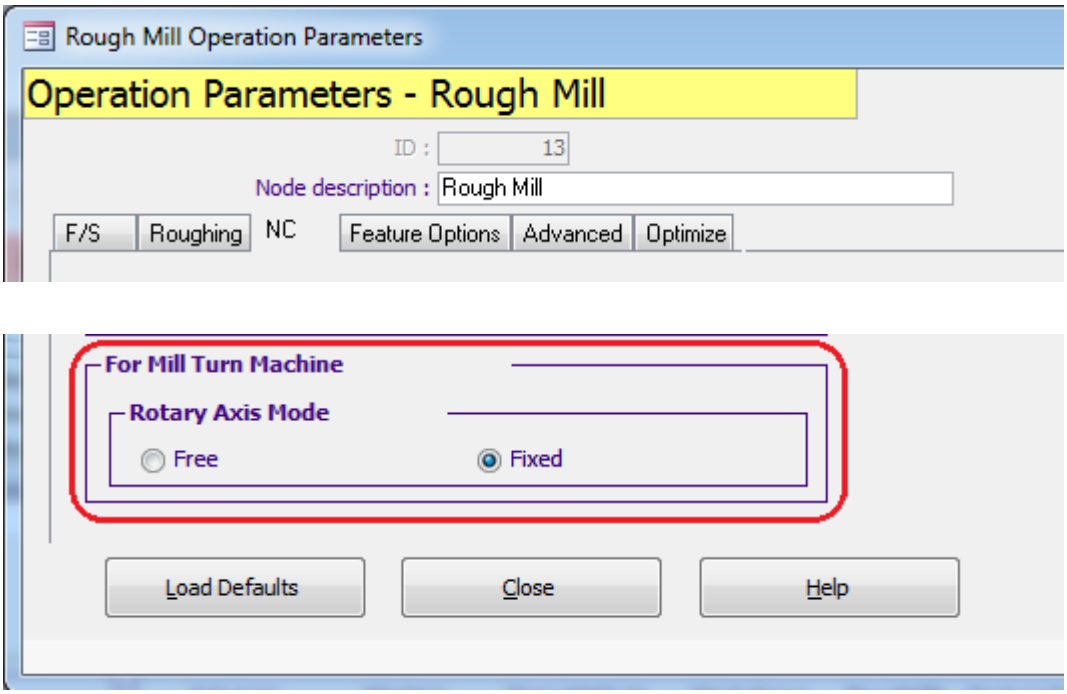
Purpose:	Enables better display of Turn Feature Extensions in the graphics area
Implementation:	The Turn Feature extensions are displayed in the graphics area in the following situations: • When interactively inserting a Turn Feature • When editing a Turn Feature • When the Feature Options tab of the Operations Parameters dialog box of a Turn Operation is active. These Turn Extensions are often used to extend the toolpath beyond the feature geometry till the WIP. They can be defined at both ends of the feature. From CAMWorks 2015 version, to enable easier interpretation of these Feature Extensions in the graphics area: 1. The Feature Extensions will be labelled as Extend1 and Extend2 in the graphics area. 2. An arrow at the end of the Feature Extension will indicate its direction.



Mill-Turn

New - Provide Rotary Axis mode options in TechDB for Mill operations

Purpose:	In TechDB's Operation Parameters Form for 2.5 Axis Milling Operations, provides options to indicate whether Milling operations will be post processed by the Fixed (XYZ) or Free (XCZ) method when Machine type is set to Mill-Turn.
Implementation:	When the Machine type is set to Mill-Turn, for post processing and simulation, it is imperative to indicate the Rotary Axis mode i.e. whether for a mill operation will be Fixed (XYZ) or Free (C-Axis "XCZ"). In earlier versions of CAMWorks, the provision to define this method is available only in the CAMWorks UI (in the NC tab of the Operation Parameters dialog box) with "Fixed" being the default method. Consequently, every time a Mill-Turn machine is used, if the user wants to assign "Free" as the default option, then he/she needs to change the setting for every Mill operation by opening the Operation Parameters dialog for each Mill operation and changing the Rotary axis mode in the NC tab. Such a task can get repetitive when Mill-Turn machine is frequently used. From CAMWorks 2015 version onwards, the provision to set the Rotary Axis mode is made available in the NC tab of the Operation Parameters Form for all 2.5 Axis Milling Operations. This enhancement will enable the user to set the preferred Rotary Axis Mode option as the default, thereby eliminating the need to change the setting for every Mill operation in the CAMWorks UI.



Rotary Axis Mode Option in Rough Mill Operation Form in TechDB