
What's New in CAMWorks 2009



CAMWorks 2009 has over 100 features and enhancements to enable you to program parts faster and more easily than ever before. A significant number of new machining options increase CAMWorks already strong manufacturing capabilities.

Click a link below or use the bookmarks.

Contents

- [General Operation](#)
- [Milling](#)
- [Multiaxis Machining](#)
- [Turning](#)
- [Mill-Turn](#)
- [Simulation / Step Thru Toolpath / Edit Toolpath](#)
- [Technology Database](#)
- [Wire EDM](#)
- [Machine Simulation](#)
- [Posting](#)

Geometric Technologies, Inc. makes no warranties, either express or implied with respect to this document. Geometric Technologies 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.



Geometric Geometric Technologies is a subsidiary of Geometric Ltd.

Copyright ©2008, 2009 Geometric Technologies, Inc. All Rights Reserved.

May 2009

General Operation

Updated - Build Info File

- Purpose** Report errors that have been corrected in the current build of CAMWorks 2009.
- Implementation** On the Start|Programs menu, select CAMWorks2009, then select Resolved CPR's. You can also open the file CW2009BuildInfo.rtf in the \lang\xxxx folder (where xxxx is the language: e.g., \lang\english).

New – 64-bit Version of CAMWorks 2009

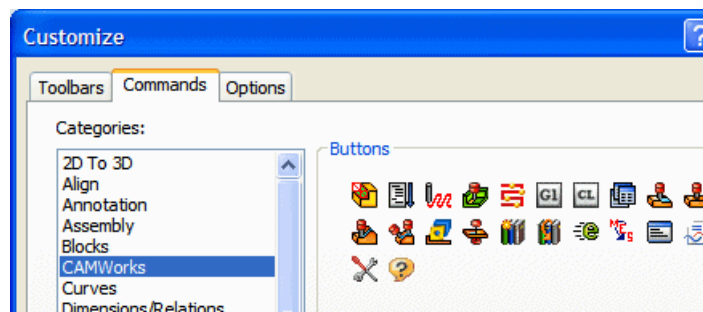
- Purpose** Provide the ability to run CAMWorks in the 64-bit version of SolidWorks 2009 and CAMWorks Solids 2009.
- Implementation** CAMWorks 2009x64 runs in SolidWorks/CAMWorks Solids 2009 on the 64-bit version of Windows XP Professional and the 64-bit version of Vista (Ultimate, Business and Enterprise editions).

New - User-defined Installation Locations for CAMWorks Program Files and Data Files

- Purpose** Provide the ability to specify the location of CAMWorks program files and data files during installation.
- Implementation** The CAMWorks 2009 setup program prompts you for the location for the program files (e.g., Program Files\CAMWorks2009) and the location for the data files (e.g., \CAMWorksData\CAMWorks2009). Data files include example and training parts/assemblies, posts, mill tooling, API files, EDM cutting conditions, machine simulators, TechDBForms.mde, TechDB.mdb and ReportDatabase.mdb. These data files require write permissions and must be installed in a folder other than Program Files if you are installing on Vista.

New - Customize CAMWorks Tool Buttons

- Purpose** Provide the ability to customize the CAMWorks toolbar and add CAMWorks tool buttons to active toolbars.
- Implementation** You can now use the SolidWorks/CAMWorks Solids Customize command to:
- add CAMWorks tool buttons to active toolbars, including the CommandManager, the Menu Bar toolbar and personal shortcut bars
 - place tool buttons on more than one toolbar
 - remove tool buttons from toolbars
 - rearrange tool buttons on toolbars
 - move tool buttons from one toolbar to another.



The Toolbar tab in the CAMWorks Options dialog box is no longer needed and has been removed.

Updated - Attribute Label Changed to Strategy

- Purpose** Change the label for Attribute to more accurately identify capabilities.
- Implementation** The Attribute label has been changed to Strategy in CAMWorks and in the TechDB.

Improved - Machine Dialog Box

- Purpose** Provide a more intuitive, easier to use interface.
- Implementation** Post Processor tab
- The Controller tab label has been renamed Post Processor to more accurately identify capabilities.
 - The active post processor is identified at the top.
 - The controller can be selected by double-clicking the post processor.
- Tool Crib tab:
- The active tool crib grid has been moved to the top, the buttons that apply to the tools have been grouped, the button to save the tool crib is clearly labeled, the available tool cribs list is at the bottom and the active tool crib is highlighted.
 - Additional changes are explained in the next item.

Improved - Tool Crib Display

- Purpose** Revise the grid for displaying tools and improve the tool crib layout to make it easier to use.
- Implementation** A grid is used to display tools from the TechDB in CAMWorks and is also used to display the current tool crib in the machine and operation dialog boxes. The following changes have been made:
- The TechDB ID has been added and the Comment field has been moved to the left in the grid.
 - The height and width of the grid on the Tool Crib tab for turning has been increased.
 - When the Add from Library button is clicked on the Tool Crib tab and the filter is set, the Tools Database form that displays is wider so additional fields can be viewed without scrolling horizontally. The form can also be expanded.

Improved - Configuration Interaction

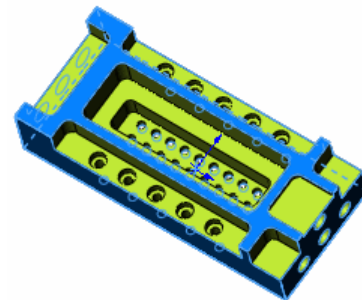
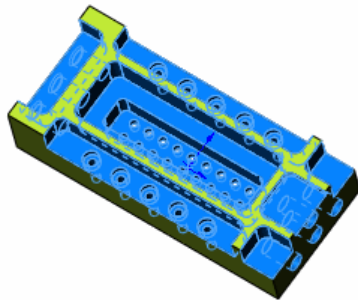
- Purpose** Modify the CAMWorks Configuration interaction.
- Implementation** New in CAMWorks 2009
- Ability to Rename configurations. By default, any new configuration name will be the same as the current Machine name. In CAMWorks 2008 the default name was the current Machine name, however it could not be renamed.
 - The following shortcut menus have been modified:
 - Configuration Manager shortcut menu commands are:
 - Manage Configurations (no change)
 - Copy Configuration (label changed for the previous Add Configuration)
 - New Configuration (new menu item that creates a new empty configuration. You select the machine to apply to this configuration after selecting the command)
 - Collapse (no change)
 - The active configuration item shortcut menu commands are:
 - Copy Configuration
 - Properties
 - Rename (added menu item)
 NOTE: The options for Show Configuration and Save Configurations are unneeded and have been removed.
 - The inactive configuration item shortcut menu commands are:
 - Show Configuration (no change)
 - Copy Configuration (no change)
 - Delete Configuration (added command)
 - Rename (added command)

New - Option to Omit Use of Configurations

- Purpose** Provide the ability to disable the Configuration Manager and configuration functions.
- Implementation** A *Use CAMWorks configurations* option has been added to the General tab in the CAMWorks Options dialog box.
- When this option is checked, the CAMWorks Configuration Manager and all configuration functions are active.
- When this option is not checked, the CAMWorks Configuration is not displayed in the tree. Also, when switching SolidWorks configurations, you will not be prompted to use a CAMWorks configuration.

New - Highlight Recognized and Unrecognized Faces

- Purpose** Highlight the faces which are not part of any feature found by AFR and the faces that are not recognized, thereby enabling easy identification.
- Implementation** The NC Manager shortcut menu in Part mode and the Feature Manager shortcut menu in Assembly mode have 2 commands:
- Highlight Recognized Faces* - the feature faces found by AFR are highlighted (shown in blue below). This command highlights only AFR features types including Holes, Multi-stepped holes and Rectangular Pockets.
- Highlight Unrecognized Faces* - the applicable faces are highlighted (shown in blue below).

**Improved - Option to Delete Un-built Features with Operations**

- Purpose** Provide a Rebuild option where features with operations that do not rebuild are deleted from the CAMWorks Feature tree.
- Implementation** In previous releases, when a feature with operations remained in the Feature tree, you had the option to Associate the feature with another feature. The *Delete un-built features with operations* option on the Update tab in the CAMWorks Options dialog box now allows these features to be deleted. This option is available for Mill, Turn and Wire EDM features.

Improved - Store Last Active Tab Location for Dialog Boxes

- Purpose** Improve the persistence in dialog boxes.
- Implementation**
- Operation Parameters dialog boxes
In previous releases, when you opened an operation dialog box, CAMWorks would remember the last tab you were on for that specific operation. Now, if you open and close an operation dialog box, the last tab that was active will be opened when you open the next operation. If you are on the Tool tab with the Holder tab active, the Holder tab will be opened when you open the next operation
 - Machine and Setup dialog boxes
When you close the Machine and Setup dialog boxes, CAMWorks now remembers the last tab, so that tab will be opened when you open the dialog box again.

Improved - Toolpath Regeneration During Rebuild

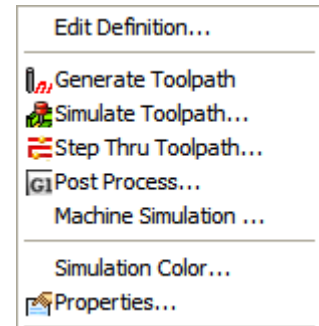
- Purpose Modify the toolpath regeneration logic so that during Rebuild, toolpaths will be regenerated reliably.
- Implementation Internal changes have been made to the rule for automatic regeneration of existing toolpaths to improve the process.
A *Generate toolpaths for operations without toolpaths* option has been added to the Update tab in the CAMWorks Options dialog box. When this option is checked, during rebuild, CAMWorks will automatically run the Generate Toolpath command for any operation that does not currently have toolpath.

Improved - Display Performance when Using OpenGL Graphics Options

- Purpose Improve performance when using OpenGL graphics.
- Implementation On the Display tab in the CAMWorks Options dialog box, when OpenGL graphics is set to Hidden Line or Wireframe, the display is smoother and faster. This improvement also applies to the display when modeless dialog boxes are open. Modeless dialog boxes allow you to interact with other windows when the dialog box is open (for example, Edit Toolpath, Step Thru, Simulate Toolpath, Stock Manager and all operation dialog boxes).

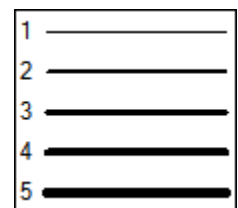
Improved - Icons Added to CAMWorks Menus

- Purpose Simplify selecting menu commands.
- Implementation Icons have been added to the shortcut menus to identify frequently used commands.



New - Set Line Weight

- Purpose Provide the ability to set the line weight of CAMWorks features, toolpaths and objects.
- Implementation The Display tab in the Options dialog box provides a *Line weight* option with a choice of 5 weights. For highlighted entities (e.g., feature definition and editing, Step Thru Toolpath and Edit toolpath), the highlighting is set to the current line weight plus 1.



New - Method to Generate XML-based Setup Sheets for Mill, Turn and Mill-Turn

- Purpose Provide an alternative method of generating setup sheets based on an XML file, which allows the setup sheet to be formatted with an HTML style sheet (.xsl) and displayed in a web browser.
- Implementation The Setup Sheet Options dialog box provides a choice to save the data and use the existing Access method or output an XML file and display as an HTML file. Options include the name of the style sheet to apply to the data in the XML file for display, the mode (Mill or Turn) and the location where the XML file will be saved. You can save and view the setup sheet when OK is clicked or simply save the setup sheet.

Improved - APT CL Output

- Purpose** Enhance APT CL support for external post processing.
- Implementation** New in this release:
- APT CL output is now supported for Turn and Mill-Turn. The APT CL button on the Post Processor tab in the Machine dialog box is now enabled for Turn and Mill-Turn.
 - A *Head/Head stock clearance* option is enabled on the Indexing tab in the Part Setup Parameters dialog box when APT CL output is selected for posted output on the Post Processor tab in the Machine dialog box. When this option is checked, an additional XYZ move is added to the end of the last toolpath before a rotary index so that the tool clears the stock. This move is added to the APT CL output, but is not graphically displayed in the toolpath. The value represents the clearance between the tool and the stock.


Improved - Toolpath Analysis

- Purpose** Compute and display the minimum and maximum XYZ values of the toolpath and the number of line and arc feed moves in the toolpath.
- Implementation** For each mill and turn operation, the toolpath analysis displays on the Optimize or Statistics tab. The minimum and maximum XYZ values will also be output to the user defined setup sheets as new variables to be displayed in the setup sheets. The NC Manager Parameters dialog box Statistics tab provides the toolpath analysis for all operations for the current part. The Statistics tab in the Operation Setup Parameters dialog box provides the toolpath analysis for all the operations in the selected Setup.

New – Option to Omit Rebuild Prompt

- Purpose** Provide an omit being prompted to Rebuild CAMWorks data when changing from one CAMWorks tree view to the other, as well as being prompted to Rebuild during a File Save.
- Implementation** When the solid model or sketches change and a CAMWorks tree view is selected, you are prompted to Rebuild CAMWorks data. If a rebuild was not performed, you would also be prompted to rebuild during File Save. The Update tab in the CAMWorks Options dialog box now includes an option in the *Prompt to rebuild* group box named *On CAMWorks tree selection and file save*. This option is available for Mill, Turn and Wire EDM features.

Improved – Collapsing and Restoring Dialog Boxes

- Purpose** Simplify collapsing and expanding dialog boxes.
- Implementation**  Minimize and Maximize buttons have been added and there are additional methods to collapse/expand the dialog boxes. To collapse a dialog box: click the Minimize button, click the X button or double click anywhere on the blue title bar. To expand a collapsed dialog box: click the Maximize button, click the X button or double click on the title bar. The expanded dialog box displays the tab that was active before it was collapsed. The Expand button in previous releases has been removed. To close a dialog box, you can click the OK/Cancel buttons, click the X button or press Alt+F4.

Improved – Option to omit warning message when Feeds & Speeds is not installed



- Purpose** Provide an option to permanently omit the warning message that displays when Feeds & Speeds is not installed.
- Implementation** In previous releases, when the *Do not perform this check again* option was checked in the warning message, the message was turned off for the current session only. This option now applies permanently.

The *Show feed and speed library warning* option has been added on the General tab in the CAMWorks Options dialog box to turn off/on the message.

Improved – Access to CAMWorks Trees

- Purpose** Change the active tree view to a CAMWorks tree when a CAMWorks toolbar command is picked.
- Implementation** When you pick a CAMWorks toolbar command, the active tree view will change to a CAMWorks tree view based on the following rules:
- Extract Machinable Features - Feature tree
 - Generate Operation Plan - Operation tree (implemented in previous releases)
 - Generate Toolpath - Operation tree
 - Simulate Toolpath - Operation tree
 - Step Through Toolpath - Operation tree
 - Post Process - Operation tree
 - Save CL File - Operation tree
 - Open TechDB - Feature tree unless Operation tree is already active, then stay on Operation tree
 - Options - Feature tree unless Operation tree is already active, then stay on Operation tree
 - Message Window - Feature tree unless Operation tree is already active, then stay on Operation tree
 - Process Manager - Operation tree unless Feature tree is already active, then stay on Feature tree
 - Insert 2.5 Axis Mill Operation - Operation tree
 - Insert Hole Operation - Operation tree
 - Insert 3 Axis Mill Operation - Operation tree
 - Insert Multiaxis Mill Operation - Operation tree
 - Insert Turn Operation - Operation tree
 - Insert EDM Operation - Operation tree
 - Create Library Feature - Feature tree
 - Insert Library Feature - Feature tree
 - Publish e-Drawing - Operation tree unless Feature tree is already active, then stay on Feature tree
 - Help - Feature tree unless Operation tree is already active, then stay on Operation tree

New - CAMWorks Library Feature

- Purpose** Implement similar functionality to SolidWorks Library features.
- Implementation** Instead of importing an entire document in another document, only some of the features and operations may need to be imported from one document to another. This is managed with the CAMWorks Library Features command. There are 2 steps:
1.  Create a library feature: Click the Add to CAMWorks Library button on the CAMWorks toolbar. Select one or more CAMWorks features from the CAMWorks feature tree that will be included in the CAMWorks library feature. Input the library feature name. Pick the folder where the library feature will be saved and click Save.
 2.  Insert a library feature into a part document: Before inserting a mill feature into a part document, you must have a .sldprt file open. Click the Insert CAMWorks Library Feature button on the CAMWorks toolbar. Select the desired library feature file. To locate the feature within the active part:
 - Select an existing mill part setup or create a new part setup to establish the Z machining direction for the feature.
 - Highlight the desired feature from the Machinable features list.
 - Reconcile the profile geometry and reconcile the end condition.

Milling

New - Import Mill Part Data into Assembly Mode

- Purpose** Provide the ability to import CAMWorks data from a part document (.sldprt) into an assembly document (.sldasm).
- Implementation** To import the part CAM data into the assembly, add the parts to the Part Manager. Right click on the part name item under the Part Manager and select Import Part Data. The CAMWorks Import Part Data dialog box displays the following options:
- Delete existing data – When selected, any existing CAM data for the selected part will be deleted from the assembly file and the new data will be imported.
 - Merge with existing data – When selected, the CAM data will be added to the current CAM data for the part.
 - Merge setups – Features and operations that are in the same Setup will be merged on both the imported part as well as with existing Setups.
- The data is copied into the assembly and not linked back to the original part. If the part contains CAMWorks configurations, the data from the Active CAMWorks configuration is imported.

Improved - Updates to AFR

- Purpose** Improve the performance of AFR to find more prismatic features.
- Implementation** Done automatically.

Improved – Creation of IFR Mill Features from Sketch Entities

- Purpose** Allow sketch entities to be graphically selected to create 2.5 Axis Mill features and provide the ability to create features from only selected sketch entities.
- Implementation** In previous versions of CAMWorks, 2.5 Axis Mill features could be created from sketches by selecting the sketch object by name. Defining 2.5 Axis Mill features from sketches has been improved so that the sketch entities can be selected graphically and features can be created from only selected entities within the sketch. This allows a single sketch to include entities for creating many different types of features. To create a feature by graphically selecting sketch entities:
- Select the *Contour Select Tool* command on the SolidWorks/CAMWorks Solids shortcut menu or check the *Contour Select Tool* option in the dialog box and pick a sketch entity. All sketch entities connected to the selected entity will be selected automatically. Note that when using the Contour Select tool, you are only able to select entities from a single sketch (i.e., to define a feature you cannot select some entities from one sketch and select additional entities from a different sketch).
 - or
 - Hold down the Ctrl key and select entities. The segments are added to the Entities selected list.

Improved – Faster Creation of Multi Surface Features

- Purpose** Improve the performance when inserting Multi Surface features.
- Implementation** In previous releases, when models contained a large number of faces and/or surfaces, the Insert Multi Surface Feature command could take considerable time to select and insert the feature. Internal changes were made to improve the performance of selecting and inserting multi surface features.

Improved – Insert Mill Setup Option to Omit Associating Entity Used


- Purpose** Provide an option to omit associating the entity used to define a Mill Setup.
- Implementation** When inserting a Mill Part Setup, if you do not want the Mill Part Setup direction to be associated to the entity used to define the setup, uncheck the *Associate* option.

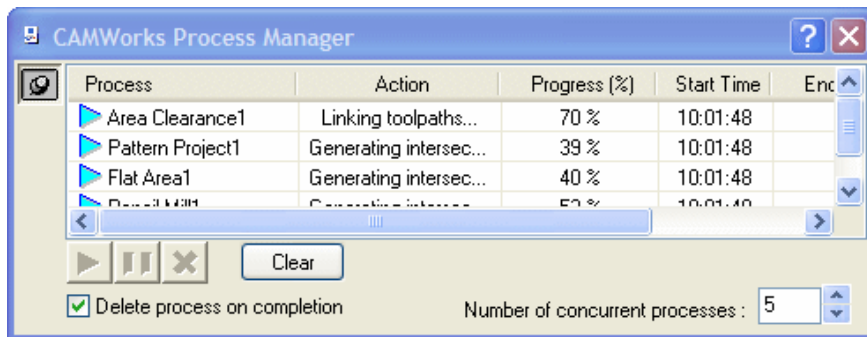
Improved – View Entity Selected to Define Mill Setup

- Purpose** Display the entity type and name used to define a user defined Mill Part Setup.
- Implementation** Right click a Mill Part Setup in the tree and select Edit Definition to view the entity selected to define the Mill Part Setup.

Improved - Multiple Process Support for Toolpath Generation

- Purpose** Provide the ability to generate 3 and 4/5 axis toolpaths as separate threads and in separate processes.
- Implementation** The generation of 3 and 4/5 axis simultaneous toolpaths often requires a significant amount of processing time and memory usage. In previous releases, you could not continue to use CAMWorks during this time. System resources, which shared memory with SolidWorks, could be exhausted when generating toolpaths requiring large amounts of memory. In this release, 3 and 4/5 axis simultaneous toolpaths run in a separate thread, which allows you to use CAMWorks while toolpaths are generating, and as a separate process, which reduces the amount of memory needed to run inside of SolidWorks. Note that the Process Manager is not supported for 3 Axis Rough and Finish operations.

 A new Process Manager can be displayed automatically when Generate Toolpath is selected or manually by clicking Process Manager on the toolbar.



The Process Manager:

- Displays the actively running processes and the current status.
- Displays the Start and End time of the calculation.
- Provides controls to Resume, Pause or Cancel a toolpath generation process and to set the maximum number of concurrent processes.

Improved - 4/5 Axis Indexing Angle Limits

- Purpose** Provide a method of defining the min and max angle limits for the 4th or 5th axis in order for CAMWorks to determine if the computed index angle is out of machine range.
- Implementation** In Part and Assembly modes, the minimum and maximum indexing limits can be defined on the Setup tab in the Machine dialog box. A *Rotation order* option allows you to change the order of rotation. The default is the 4th axis rotates first, then the 5th axis. For some machines and specific part cases, the order of rotation must be changed to rotate the 5th axis first, then the 4th axis. This setting can be applied to all existing setups.
- A *Use Global rotation order* option is also on the Indexing tab for each Setup. When checked, this option is set to the *Rotation order* that is specified on the Machine Setup tab and the *Rotation order* option is disabled.
- If the 4th or 5th axis computed index angle is out of the machine range, CAMWorks first computes an alternate solution and second, if both solutions are out of range, reports that the computed angle is out of range.



An exclamation mark displays on a Setup item when CAMWorks is unable to resolve indexing angles that are out of range. To understand what the error means, right click the Setup and select What's Wrong. The defaults for the index limits can be set on the Mill Machine Parameters form in the TechDB.

Improved – Generate Predrilled Hole at the Leadout Position for 2 Axis Rough and Contour Operations

Purpose Provide an option to generate a predrilled hole at the leadout position of the 2 axis rough and contour operations.

Implementation *Retract Hole* has been added to the existing Retract methods on the Feature Options tab. When Retract Hole is selected, a hole will be created at the XYZ position of the last point of the toolpath. The retract hole operations will be based on the Strategy defined under the Entry Hole option. When Z Rapid is active, the tool will rapid in Z as is the current behavior.

Updated - Entry Point Label Change

Purpose Change the *Entry point* label on the Entry/Retract tab for Area Clearance and Flat Area operations to more accurately identify the option.

Implementation The *Entry point* label has been changed to *Entry hole*.

Improved – Adaptive Roughing Toolpath Computation Speed

Purpose Improve toolpath computation speed.

Implementation Done automatically.

Improved – Adaptive Roughing *Min Corner Radius* Option

Purpose Allow a user defined radius for the smallest trochoidal move that can be generated. This can restrict the tool from machining slots that are only slightly wider than the cutter diameter.

Implementation For the Adaptive Roughing pattern, set a value for the *Min corner radius* option in the Limits group on the Area Clearance tab.

Improved – Adaptive Roughing Support for Rest Machining

Purpose Support rest machining.

Implementation For Area Clearance operations, when Adaptive is selected for the Pattern on the Pattern tab, the Rest tab is enabled.

Improved – Adaptive Roughing Option to Disable Adaptive Stepdown

Purpose Provide an option to disable adaptive stepdown.

Implementation When the *Adaptive stepdown* option is checked on the Area Clearance tab, the Z stepdown level in Adaptive roughing is based on the Max cut amount, Cut amount and Min cut amount. The first cut depth is equal to the Max cut amount, where the area of the entire level is machined. The next Z level is above this level and only machines the material that was not removed by the level below it. The amount that the tool moves up in Z adapts to the shape of the part and is based on the Cut amount, but will not be less than the Min cut amount. When this option is not checked, the amount that the tool moves up in Z is constant and equal to the Cut amount.

Improved – Constant Stepover Initial Toolpath Pass

Purpose Improve the quality of the initial toolpath pass.

Implementation Done automatically.

Improved – Constant Stepmover Option to Specify Offset Curve Based on Contain Area Only

Purpose	Provide an option to specify the Offset curve based on the contain area only.
Implementation	When a Constant Stepmover operation contains both contain and avoid areas, the resulting toolpath pattern is based on the offsets from the contain and all the avoid areas. An option has been provided to define the toolpath pattern from the contain area only. To define the pattern from the contain area, select the <i>Offset curve: Contain Only</i> option on the Constant Stepmover tab.

Improved – Area Clearance Option to Disable Optimization of Z Levels

Purpose	Provide an option to disable the optimization of Z levels in the Area Clearance patterns: Pocket Out, Lace and Pocket in – Core.
Implementation	For Area Clearance operations, an <i>Optimize Z levels</i> option has been added to the Depth Parameters group on the Area Clearance tab. When this option is not checked, the Z levels are always a whole multiple of depth Cut amount. It may be possible that this will result in Z levels that are coincident with near planer surfaces, causing fragmentation of the toolpath.

Improved – Area Clearance Spiral and Ramp Entry Moves

Purpose	Improve the number of scenarios where Spiral or Ramp entry moves can be used successfully.
Implementation	Done automatically.

Improved – Limit Depth Option for Area Clearance, Flat Area and Z Level

Purpose	Provide an option to limit the machining depth based on the Z value of the entry hole.
Implementation	A <i>Limit depth</i> option has been added on the Entry/Retract tab for Area Clearance, Flat Area and Z Level operations. When checked, the roughing toolpath depth will be limited by the Z value of the entry hole position.

Improved – Z Level Option to Specify Entry Hole Position

Purpose	Provide the ability to specify an entry hole position.
Implementation	On the Entry/Retract tab of the Z Level operation, check the <i>Use entry holes</i> option and select the entry point positions.

Improved – Z Level Horizontal Arcs Option

Purpose	Provide the ability to specify horizontal leadin and leadout arc moves.
Implementation	On the Entry/Retract tab of the Z Level operation, check the <i>Horizontal arcs</i> option.

Improved – Rapid Plane Definition for 3 Axis Milling

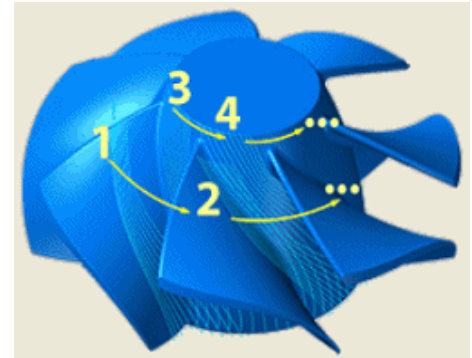
Purpose	Provide additional control in defining the Rapid plane position.
Implementation	The definition of how the options function for defining the Rapid plane in 3 axis operations has changed. Previously, regardless if the Rapid plane was defined from the Top of Feature, Setup Origin, Clearance Plane, or Top of Stock, CAMWorks would automatically compute a safe Z value that cleared the top of the stock or top of the feature by an amount equal to the Skim clearance on the Entry/Retract tab. If this safe Z value was greater than the Z position based on the Rapid plane method and distance value, the safe Z value was used to define the Rapid plane. When the new option "Auto safe Z plane" is not active, the Rapid plane will ignore the Z safe position and directly use the specified rapid plane. This allows the rapid plane to start below the top of the feature or stock. By overriding the Z safe plane, the tool may collide with the stock or part. By default this new option will be active for existing toolpaths.

Multiaxis Machining

Improved - Rotate and Copy Toolpaths Around an Axis

Purpose Provide the ability to rotate and copy toolpaths around an axis.

Implementation This feature can be used to create full rotation symmetric operations (for example, impeller machining). Options are on the Advanced tab in the Operation Parameters dialog box.

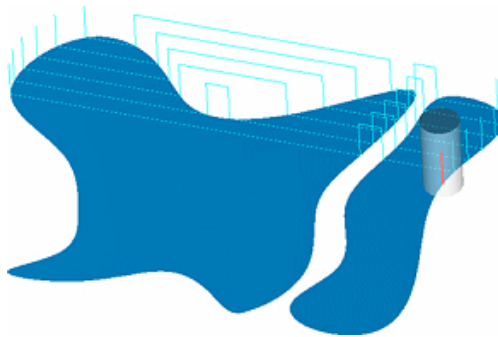


New - Machine by Lane or Region

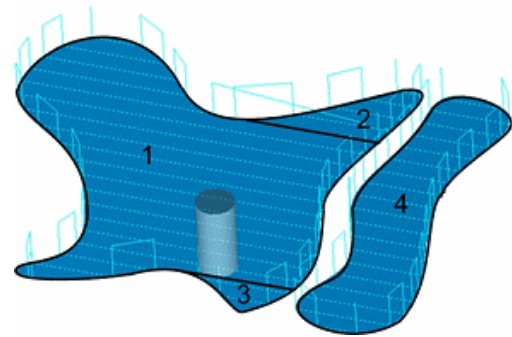
Purpose Provide the ability to machine surfaces continuously or by regions

Implementation The generated toolpath usually has a topology of multiple contours (lanes) on the surfaces. When the toolpath is generated on many zones, it might be preferable to machine all the regions independently. This option on the Pattern tab defines whether to follow the machining by lanes or by regions.

Machine by lane



Machine by region



New - Max Projection Distance Option for Curve Projection

Purpose Provide an option to specify a maximum distance the projection curve can be above the surface.

Implementation The *Max projection dist* option on the Pattern tab defines the maximum distance between the projection curve and the surface. For example, if the value is 0.1mm, then the projection curve can be 0.1mm away from the surface.

New - Slow and Safe Toolpath Creation

Purpose Provide more accurate analysis of complex surface topology to generate toolpaths.

Implementation The creation of surface contact paths is done by analyzing the surface patches and slicing it. The toolpath topology may become very complex due to slicing (for example, parallel to curve and surface patches are very large) and there are cases where the surface contact paths cannot be constructed safely. When the *Slow and safe toolpath creation* option on the Finish tab is checked, a finer grid (based on the max. step over value) is applied for the initial analysis of surface patches, which delivers slow but safe results for surface contact points.

New - Surface Edge Merging Distance

Purpose Provide a merge distance to further control merging surface paths.

Implementation Surface paths are created on individual surfaces, then they are merged together into longer surface paths. The decision about merging is based on a merge distance. If all surface paths on a toolpath slice are merged, CAMWorks calculates whether a closed surface path can be built by connecting the start to the end based on the merge distance value. The Distance option on the Links tab specifies the limit value for the merge distance as a value or a percentage of the tool diameter. All surface paths that are within a closer distance than this value are merged.

New - Point Tool to Rotary Axis Option for 4 Axis

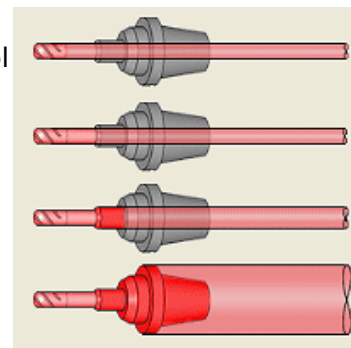
Purpose Provide an option in 4 axis machining to make the tool cross the rotary axis.

Implementation The *Point tool to rotary axis* option on the Axis Control tab is used for 4 axis machining, which has the option to lock the 5th axis to an angle. In most cases users want to make tool axis cross the rotary axis also. If this option is checked, the calculation will be done like the tool will be tilted with a fixed angle.

New - Gouge Checking Option to Extend Tool to Infinity

Purpose Provide an option to extend the tool geometry to infinity for more accurate gouge checking.

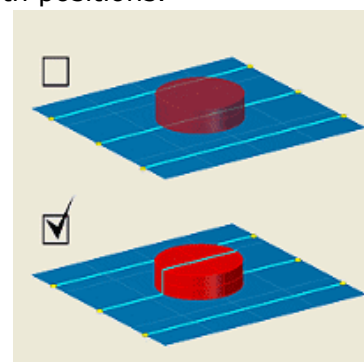
Implementation The *Extend tool to infinity* option on the Gouge Checking tab turns on and off the extension of tool geometry to infinity. If checked, then the tool, arbor and holder will be considered with the given length for collision checking purposes. For several reasons, it might be desirable to extend the last used portion of the tool with a cylinder to infinity. This will help the collision checking system to detect all collisions.



New - Gouge Checking Option to Check Gouges Between Positions

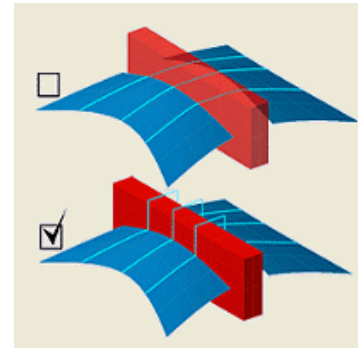
Purpose Provide an option to check gouges between toolpath positions.

Implementation When the *Check gouges between positions* option is selected on the Gouge Checking tab, collision checking is performed between toolpath positions. Then, the 5 axis sweep move from one position to the next position is used to check for collisions with the surface to cut and the surfaces to avoid. To get correct toolpaths and to avoid gouges, always select this option. If this option is not selected, collision checking is performed at every end point of the toolpath.



New - Gouge Checking Option to Check Link Motions for Collision

- Purpose** Provide an option to check link motions for collision.
- Implementation** The *Check link motions for collision* option on the Gouge Checking tab allows enabling/disabling the check of link moves for collision during toolpath generation.

**New - Gouge Checking Option for Tilt Tool Away With Max Angle Strategy**

- Purpose** Provide an option to use lead/lag angle and side tilt angles for collision checking with the Tilt Tool Away with Max Angle Strategy.
- Implementation** When Tilt Tool Away with Max Angle is selected for the Strategy on the Gouge Checking tab, *Use Lead/Lag and Side Tilt Angles* can now be selected for the Angle definition. CAMWorks determines whether to use the side or lead and lag direction for tilting or a combination of both. You can set an angle range in which CAMWorks can tilt the axes as well as numerous parameters to control gouge checking.

New - Stock Collision Checking

- Purpose** Provide an option for collision checking when using stock.
- Implementation** When Rest machining is enabled on the Rest tab, the *Stock collision checking* option on the Gouge Checking tab allows you to add the shaft, arbor and/or holder to stock consideration during toolpath generation.


Improved - Links Between Passes

- Purpose** Provide an option to use different method of linking multi pass toolpaths for passes greater than and less than a specified value.
- Implementation** In CAMWorks 2008 there were controls that would define pass linking, but this control applied to all passes regardless of the distance between them. In this release, a link threshold has been added so that a different method of linking can be applied for passes that are less than the threshold distance versus those that are greater than the threshold distance. The Link type and Entry/Exit method that was defined in CAMWorks 2008 represent the control over passes that are less than the threshold value.

New - Define Contain Areas

- Purpose** Provide an option to contain the toolpath within a user specified area.
- Implementation** The contain area is defined by selecting the Insert Contain Area command on the shortcut menu for Multiaxis Mill operations. The Curve Wizard: Select Geometry dialog box displays to select a sketch, model face or model edges to define the perimeter shape of the contain area.

New – Area Roughing

- Purpose** Provide functionality to more easily create rough and floor finish toolpaths for impellers and blisks.
- Implementation** When you check the Area roughing option on the Roughing tab and click the  button, the Area Roughing dialog box displays the options that can be set for impeller and blisk floor finishing and roughing, and for slots on cylinders. These options can be used to create roughing and finishing toolpaths and machine impellers with or without a splitter.

Improved – Multiaxis Stepper

- Purpose** Allow the stepover to be calculated by specifying a scallop height.
- Implementation** In a Multiaxis operation, if the tool includes a corner radius, you can specify the *Max scallop* on the Pattern tab to automatically compute the stepover value based on a defined scallop height.

Turning

Improved - Updates to AFR

Purpose	Implement enhanced turning AFR for improved recognition of ID features.
Implementation	Done automatically.

Improved - Cutter Compensation Support for Turn Operations

Purpose	Provide the ability to generate CNC compensation codes in the G-code program for additional operations.
Implementation	<p>In previous releases CNC compensation codes could be output only for Finish Groove operations.</p> <p>In this release, a CNC Compensation On/Off option has been added on the main tab for the following operations:</p> <ul style="list-style-type: none"> • Turn Rough (With Canned Cycles) • Turn Finish • Face Rough (With Canned Cycles) • Face Finish • Bore Rough (With Canned Cycles) • Bore Finish • CutOff

New - Define Turn Stock from 2D WIP Sketch

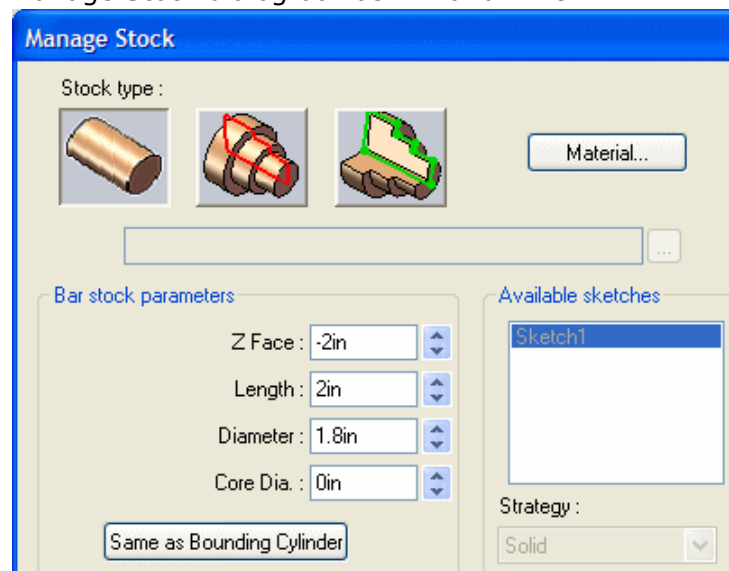
Purpose	Provide a method to create an external 2D WIP sketch that can be used to define a revolved stock shape.
Implementation	After generating toolpaths, select Save WIP Model on the Stock Manager shortcut menu and select the operations you want CAMWorks to use to calculate the WIP. To save, select <i>WIP Sketch Files (*.cwtwip)</i> for the file type.



To use the WIP model to define Turn stock, select the WIP option on the Stock tab in the Stock Manager dialog box, then select the file.

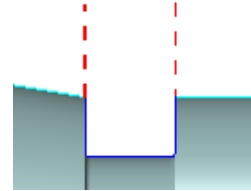
Improved - Manage Stock Dialog Box

Purpose	Provide the same format for the Manage Stock dialog box as Mill and Wire EDM.
Implementation	The Manage Stock dialog box is now similar to the dialog boxes in Mill and Wire EDM.



Improved - Feature Extends Display

- Purpose** In the Insert and Edit Turn Feature dialog boxes, provide the ability to display both Extend conditions at the same time and view both Extends on the part model.
- Implementation** When either Extend 1 or Extend 2 is selected, both Extend conditions display in the dialog box and both Extend segments display on the part model. The currently selected Extend displays as a bold red dashed line. The inactive Extend displays as a dashed red line. The Extend segments will display as dashed lines on the part model when the feature is displayed in the Feature or Operation tree.

**Improved - Delete Multiple Feature Segments**

- Purpose** In the Insert and Edit Turn Feature dialog boxes, provide a method to delete multiple feature segments using the Ctrl and Shift keys.
- Implementation** In previous releases, multiple segments could be selected/deselected using the Window Selection option. You can now select multiple segments in the dialog box. Select multiple segments individually by holding down the Ctrl key while you select each segment. Select an entire group of segments by selecting the first segment, holding down the Shift key, then clicking on the last segment of the group.

Improved - Operation Parameters Dialog Box Changes

- Purpose** Modify the main operation tab and the Advanced tab to provide consistency across operations, for clarity and to make room for new options.
- Implementation** The following changes have been made on the applicable main operation tabs:
- Output group box has been renamed *Program point*.
 - The Without compensation/With compensation option has been combined into the *Feature geometry* option in the Program point group box. This option applies to non-groove tools. When checked, CAMWorks will generate the toolpath as if the tool nose radius is 0.00. This will effectively create CL data that is the same as the feature geometry itself. Under most situations this option will not be used unless outputting canned cycle format for roughing operations. For most turning operations this option will be disabled and the actual program point is defined on the Posting tab. When this option is not checked, the program point is defined on the Posting tab. This is the most common setting and will be used almost exclusively for finish toolpaths and roughing when not using canned cycle output.
 - The Driving point option has been moved to the Program point group box, the label has been changed to *Groove tool* and the radio buttons have been changed to a drop down list.
 - The Canned cycle output option has been moved to the Program Point group box.
 - The CNC compensation On/Off option is in the Program point group box.
- In the Z limits section on the Advanced tab:
- *Minimum Z limit* has been renamed *Z start*
 - *Maximum Z limit* has been renamed *Z end*
 - The position of the options has been reversed (i.e., *Z end* is on the left and *Z start* is on the right).

Improved - Reverse Direction of Turn Setups

- Purpose** Allow the Z direction of Turn Setups in the Operation tree to be reversed.
- Implementation** The Origin tab in the Operation Setup Parameters dialog box now contains a *Reverse direction* option. The default can be set in the TechDB on the Spindle/Chuck tab on the Turn Machine Parameters form.

Improved – Negative Allowances for Turning Operations

Purpose	Implement support for negative X and Z machining allowance for all turning operations.
Implementation	In order to machine parts to specified tolerances and in situations where a part is used as part of a mold or die, the allowance or stock left on a turn part may need to be negative. The Radial (X) and Axial (Z) allowances can now be set to a negative value on the main operation tab for Face Rough and Finish, Turn Rough and Finish, Bore Rough and Finish, Groove Rough and Finish, and Cut Off operations.

Improved - Reverse Direction of Rough Turn and Face Rough Toolpaths

Purpose	Provide an option to reverse the toolpath direction for Rough Turn and Face Rough.
Implementation	Previous versions of CAMWorks supported reversing the direction for Turn Finish, Face Finish, Bore Rough, Bore Finish and Threading operations. Support has been extended to Turn and Face Rough and a <i>Reverse</i> option has been added to the Rough Turn and Face Rough tabs.

Improved - Turn Thread Output To Tip or Center

Purpose	Provide an option in turn threading so that the toolpath output is either through the tool nose center or the tool nose tip
Implementation	The Thread tab in the Operation Parameters dialog box contains a Program point option to select Tool nose tip or Tool nose center.

Improved - Grooving with Round Insert

Purpose	Allow grooving with a round insert.
Implementation	All groove operations including Rough and Finish Groove of every feature type allow the selection of a Round Insert tool/holder within each operation. In the TechDB tool selection for Groove Operations, both groove and round inserted tools are supported.

Improved - Incremental Grooving Feed Plane

Purpose	For Rough Groove and Cutoff operations, provide an option for an incremental feed plane.
Implementation	The <i>Feed dist.</i> option on the NC tab in the Operation Parameters dialog box defines a distance where the tool will rapid to before changing to feed motion.

Improved - Support Diameter and Length Option for Bore and Face Finish

Purpose	Provide the ability to generate a bore finish and face finish toolpath based on a user supplied diameter and length.
Implementation	The same functionality supported for Turn Finish has been implemented for Bore Finish and Face Finish.

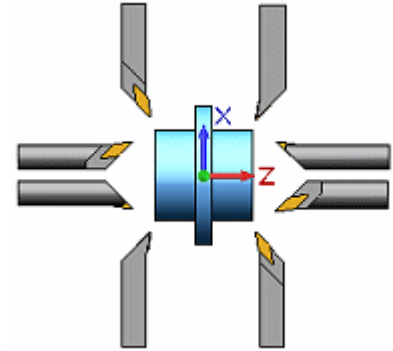
Improved - Automatically Set Turn Approach/Retract with Gouge Check

Purpose	Provide an option to generate gouge free Approach/Retract strategy with minimum user interaction.
Implementation	The Auto option has been added to all Turn operations (Turn Rough/Finish, Bore Rough/Finish, Rough Groove/Finish, Face Rough/Finish, Center Drill, Drill, Cutoff and Thread). When checked, the approach and retract moves will be calculated automatically and will be gouge free.

New - Display of Left and Right Hand Turn Tools in Step Thru and Simulate Toolpath

- Purpose** Display turn holders properly in Step Thru Toolpath and Simulate Toolpath based on the definition of right and left hand holders.
- Implementation**
- The Boring Bar and Holder pages on the Tool tab in the Operation Parameters dialog box display the hand of the tool (Left or Right) in a drop down list. The default value is set based on the value for *Hand of cut* on the Tool Holders and Holders - Boring Bar forms in the TechDB.
 - The Holder page includes a new option to see and set the *Shank thickness*. The default value for the shank thickness is defined on the Tool Holders and Holders - Boring Bar forms in the TechDB.
 - The Boring Bar and Holder pages include a new graphic that changes based on the selection of a Left or Right hand holder.
 - For Groove inserts, a *Groove driving point* group box has been added and the *Primary driving point auto* and *Primary is* options have been moved into this group box.
 - All the turn insert pages include an option to see and set the *thickness* of the insert.
 - The tool and holder display in Step Thru and Simulate Toolpath has been updated to reflect the Left or Right hand setting and the thickness of the insert and holder.
 - The defaults for the new tool parameters defined above can be set in the TechDB.

Right Hand Holder Orientations



Improved - Back Turning Support

- Purpose** Allow turning toolpaths to be generated where the orientation of the insert is opposite of the typical holder orientation (as shown in the figure).
- Implementation** Set the insert angle for non groove tools to the desired angle, verify the Program point selection, and select Reverse toolpath.



Mill-Turn

New - Support for 5 Axis Simultaneous Machining

- Purpose** Add support for 5 Axis simultaneous machining in Mill-Turn mode.
- Implementation** In previous releases, Multiaxis operations in Mill-Turn supported only 3 and 4 axis motion. The number of axis is no longer be limited to 3 and 4 axis mode in Mill-Turn mode, but will observe the multiaxis license level.


Improved - Wrapped Feature Toolpath Diameter

- Purpose** Provide an option to compute a wrapped toolpath at either the top or bottom of the feature.
- Implementation** CAMWorks computes a flat layout length for the wrapped toolpath calculation. In previous releases, CAMWorks based the computation on the diameter at the bottom of the feature. In the Insert Wrapped Feature dialog box, the *Wrap Dia.* option now allows you to define whether the flat layout will be calculated based on the diameter at the top or bottom of the feature.

Simulation / Step Thru Toolpath / Edit Toolpath


Improved - Toolbar Button to Display XYZ Coordinates during Simulation

Purpose Provide a toolbar button to enable/disable the display of cutter coordinates in the Message Window during Simulate Toolpath for Mill, Turn, Mill-Turn and Wire EDM.

Implementation  A button has been added to the Simulate Toolpath toolbar that toggles the cutter coordinate display on/off. When selected, the Message Window will open. This button provides the same functionality as the *Cutter coordinates in Message Window* option on the Simulation tab in the CAMWorks Options dialog box.

Improved - Simulate Toolpath Toolbar Button to Display Simulation Options

Purpose Provide a toolbar button to open the Simulation tab in the CAMWorks Options dialog box for Mill, Turn, Mill-Turn and Wire EDM.

Implementation  A button has been added to the Simulate Toolpath toolbar that opens the Simulation tab in the CAMWorks Options dialog box.

Improved - Display Radial or Diameter Values in Step Thru and Simulation for Turn and Mill-Turn

Purpose For Turn and Mill-Turn parts, allow the X coordinate to display as either a radial or diameter value during Step Thru.

Implementation In previous versions of CAMWorks, the output was always radial. When viewing the coordinates, they did not match the G-code output for some machines that require diameter values.



A button has been added to the Step Thru Toolpath and Simulate Toolpath toolbars that toggles between Radial and Diameter. When the button is not pressed, the X coordinates will be output in Radial mode. When the button is pressed, coordinates will be output in Diameter mode. For Simulate Toolpath, the Cutter coordinates button must be pressed in order to display this button.

Improved - Display Tip or Center Coordinates in Step Thru and Simulation for Turn and Mill-Turn

Purpose For Turn and Mill-Turn parts, allow the XZ coordinates to be output to the tool center or the theoretical tool tip during Step Thru.

Implementation In previous versions of CAMWorks, coordinates were output to the center of the insert radius.



A button has been added to the Step Thru Toolpath toolbars that toggles between Tip and Center coordinate display. When the button is not pressed, the XZ coordinates will be output to the insert radius center. When the button is pressed, coordinates will be output to the tool tip. For Simulate Toolpath, the Cutter coordinates button must be pressed in order to display this button.

Improved - Simulation Reliability with STL Stock

Purpose Improve reliability of simulation with STL stock.

Implementation Internal changes have been made. No user action is required.

Improved - Turn Boring Bars Display as Round Shapes

Purpose Display turn boring bars as a round shape in Step Through and Simulation.

Implementation In previous releases, boring bars were displayed as a rectangular cross section, which did not reflect the true shape of the holder. Boring bars now display as a round shape.

Improved - Unlimited Number of Toolpath Segments to Display for Step Thru and Edit Toolpath

- Purpose Increase the number of toolpath segments to display during Step Thru Toolpath and Edit Toolpath.
- Implementation The maximum number has been increased from 32000 to an unlimited number.


Improved - Store Values for Number of Segments and Step Increment for Step Thru Toolpath and Edit Toolpath

- Purpose Store values for *Number of Toolpath Segments to display* and *Multiple Step Increment* for Step Thru Toolpath and Edit Toolpath.
- Implementation The maximum number of toolpath segments and the multiple step settings are now saved and will be restored when the toolbar and dialog box are reopened. These values will be the same for both Step Thru and Edit Toolpath.

Improved - Time to Start Edit Toolpath for Very Large Toolpaths

- Purpose Increase the speed for starting Edit Toolpath.
- Implementation Starting Edit Toolpath for very large toolpaths is now faster.

Improved - Option to Select Part in Assembly for Turbo Mode Simulation

- Purpose In Assembly mode, when multiple parts/stocks exist, provide an option to select which part/stock to perform the simulation when running in Turbo mode.
- Implementation  A new button has been added to the Toolpath Simulation toolbar when in Turbo mode and multiple parts exist. When clicked, a list of parts/stocks displays for selecting the stock.

Improved - View All Parts when using Compare in Simulation

- Purpose When selecting Show Difference in Tool mode, if more than 1 part exists allow all parts to be viewed, not just the first part.
- Implementation When the Show Difference button is clicked on the Toolpath Simulation toolbar, all parts remain displayed.

Technology Database

New - Set Default Thickness for Inserts

- Purpose** Provide ability to define defaults for Insert thickness.
- Implementation** A default insert *thickness* can be defined on each turn insert form including Diamond, Groove, Thread, Round, Hexagon, Square, Triangle and Trigonal. This parameter defines the thickness of the insert, as measured normal to the XZ turning plane.

New - Tool Holder Definition Simplified and New Parameters added

- Purpose** Revise the Tool Holder form for clarity and add support for new Shank thickness parameter.
- Implementation** Four fields have been added to the turn Tool Holders form: Holder edge, Insert type, Shank thickness and Hand of Cut. The Holder edge and Hand of Cut fields were moved from other forms. *Shank thickness* is a new parameter that defines the thickness of the holder, as measured normal to the XZ turning plane.

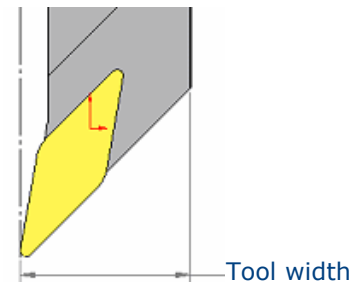
New - Tool Holder - Boring Bar Definition Simplified

- Purpose** Revise the Tool Holder-Boring Bar form for clarity.
- Implementation** Four fields have been added to the turn Tool Holder - Boring Bar form: Holder edge, Insert type, Minimum cut diameter and Hand of Cut. These fields were moved from other forms.

New - Tool Width Parameter on Turn Tool Assemblies Form

- Purpose** Revise the Turn Tool Assemblies form to change labels, modify existing fields and add support for the Tool width parameter.
- Implementation** In the Turn Holders drop down list, the Default label has been changed to Holder. The Hand of Cut field has been removed. The value is set based on the selected Holder or Boring Bar Hand of Cut field.

A Tool Width parameter has been added to define the total width of the tool from the back of the holder/boring bar to the tip of the insert. The value also defines the minimum diameter that can be machined by the boring bar in an ID feature. The CAMWorks toolpath algorithm considers this value and will not generate any toolpaths where the resulting pass would be less than this value.



Improved - TechDB Definition of Protrusion for Tap Tools

- Purpose** Ability to set the value for Protrusion for Tap tool definitions in the TechDB.
- Implementation** A *Protrusion* parameter has been added to the Tools Database - Tap form.

Improved - Change Holder - Default label

- Purpose** Change Holder - Default to Tool Holders for clarification.
- Implementation** The *Holder's Default* label has been changed to *Tool Holders* in the Tool Holder form. The associated labels on other forms have been changed including the Default Holders label in the tree, the Default Holders label on the Turn Tooling menu and the Default option for Holder Type on the Turn Tool assemblies form.

***Improved* - Update TechDB Data for Mill Tooling and Mill Threading**

Purpose Update the Tool libraries for mill tooling and thread milling conditions.

Implementation The TechDB tooling data has been updated.

***New* – Turn Tool Crib Parameter**

Purpose Add a field to assist in tool loading.

Implementation The *Orientation* field defines the orientation of the holder when it is in the turret.

Wire EDM

New - Multiple Glue Stops

- Purpose** Provide the ability to define multiple glue stops.
- Implementation** A Glue Stops tab has been added to the 2 Axis and 4 Axis Contour dialog boxes. Up to 8 glue stops can be defined. The location method and length of each glue stop can be specified. A leadin and leadout can be defined for each glue stop.

Improved – Copy Features

- Purpose** Provide the ability to copy machinable features.
- Implementation** EDM features can be copied by pressing the CTL key, then clicking on the item you want to copy and dragging the item to a new location.

New - Set Defaults for NC Program File Path and File Extension

- Purpose** Allow defaults to be defined for the path for the NC program file and the file extension.
- Implementation** The Settings tab in the EDM Machine Setup program (CWEdmMachinesSetup.exe) in the CAMWorksData\CAMWorks2009\EDM\Posts folder has options to set the path for the NC program file and the file extension.
- In CAMWorks the Posting tab in the EDM Settings dialog box has an *NC File Path* option to change the default folder to store the NC program. Optionally, the path can be saved as the default.

New - User Comments for Individual Operations

- Purpose** Provide the ability to specify and output user comments for individual operations in the G-code.
- Implementation** A User Comments tab has been added to the 2 Axis and 4 Axis Contour dialog boxes. You can enter comments for the rough and skim passes. If the *Output user comments* option is selected, the comments are output as variables that can be put anywhere in the G-code. Post processor customization is required to output the comments to the G-code.

Improved - Work Offsets

- Purpose** Provide the ability to specify a Work offset for each operation.
- Implementation** The Work offset can be defined on the Posting tab in the 2 Axis and 4 Axis Contour dialog boxes.

New - Toolpath Preview

- Purpose** Allow toolpath changes to be previewed when an operation dialog box is open.
- Implementation** A Preview button has been added to the 2 Axis and 4 Axis Contour dialog boxes that allows you to view the effect of current operation parameters on the toolpath. When you click this button, the toolpath for the current operation is calculated and displayed immediately. This allows you to preview the toolpath for the operation based on the current operation parameters. When you preview a toolpath, CAMWorks temporarily collapses the dialog box to show only the title and menu bars. To restore the dialog box, click the X button in the top right corner. The dialog box expands and displays the tab that was active before the Preview.

Improved - Machine Stop Control

- Purpose** Provide the option to set the Machine stop type per operation.
- Implementation** The Stop type can be defined globally for all operations or it can now be defined differently for each operation on the Leadin/Out tab in the 2 Axis and 4 Axis Contour dialog boxes.

New - Define Custom Post Processor Variables

- Purpose** Provide the ability to define custom variables for 2 and 4 axis Contour operations and EDM Settings that can be output in the NC program.
- Implementation** Files can be created to define custom post variables that can be read by the post processor and output in the NC program. Variables can be defined for 2 Axis Contour operations, 4 Axis Contour operations and for EDM Settings. The variables display on the Adv Posting tab in the operation dialog boxes and the EDM Settings dialog box.

New - Documentation for Post Commands and Variables

- Purpose** Provide an EDM Post Processor reference.
- Implementation** The commands and variables are documented in the EDM Post Reference help (available on the Help menu in the UPG) and the CAMWorks Wire EDM Post Processor Writer's Reference (EDM_Post_Reference.pdf), which is installed in the UPG folder.

New - Support for Post Scripting API Functions

- Purpose** Provide support for API functions for Visual Basic scripting (VBScript).
- Implementation** The API functions are documented in the EDM Post Reference help (available on the Help menu in the UPG) and the CAMWorks Wire EDM Post Processor Writer's Reference (EDM_Post_Reference.pdf), which is installed in the UPG folder.

New - Assign Machine Stops Along Rough Cut Toolpath

- Purpose** Allow machine stops to be identified at specified locations along the toolpath.
- Implementation** While machine stop codes can be output automatically, there are situations where you may need to place a machine stop code at a location that cannot be modified. The Posting Attribute tab has been added to the 2 Axis and 4 Axis Contour dialog boxes. This tab allows you to assign Machine Stop attributes to the feature geometry. CAMWorks then applies these to the Rough Cut toolpath while generating the toolpath.

Machine Simulation

Improved - Opens in Full Screen

Purpose Open Machine Simulation in full screen mode when started.
Implementation Done automatically.

Improved - Ability to Tab back to SolidWorks


Purpose Provide the ability to Tab to SolidWorks when Machine Simulation is open.
Implementation You can Tab back to SolidWorks and pick items in the tree. No changes to CAMWorks data can be made in SolidWorks (e.g., edit operations).

Improved – Positioning the Part on the Machine

Purpose Provide the ability to select a point to position a part on the machine and retain the point in the part.

Implementation Instead of selecting the XYZ offsets when you open Machine Simulation, you can pick a point in CAMWorks to define the position of the part. The position is stored and reapplied automatically when you reopen Machine Simulation.

To position the part on the machine:

1. Open Machine Simulation.
2.  Click the button to the right of the Machine. The Select Point dialog box displays.
3. Graphically set the position using one of the following selection methods: Vertex selection, arc edge selection, graphical selection of a single point or arc/circle entity from a sketch, or pick a sketch from the SolidWorks tree, which will extract all point and/or arc/circle entities from sketch.
4. In Machine Simulation, click the Update button to the right of the offsets.

New - User Interface

Purpose Provide an enhanced user interface.
Implementation Done automatically.

Posting

Post customization is required in order to use new commands and variables. These commands and variables are not supported in previous versions of CAMWorks or in any ProCAM product.

***New* - Post Processor Output for Node Names**

Purpose

Provide post processor variables that report the various node names associated to each toolpath.

Implementation

Internal post processors can be customized to output the following node names during post processing: Part name-Part configuration, Setup name, Setup description, Feature name, Feature description, Operation name.

In the APT CL output, keywords and associated values are output for each of the node names.

TLP_PART_NAME

Type CHARACTER

Usage

Stores the toolpath part name. Available in Mill, Turn and Mill/Turn.

TLP_PART_DESC

Type CHARACTER

Usage

Stores the toolpath part description. Available in Mill, Turn and Mill/Turn.

TLP_SETUP_NAME

Type CHARACTER

Usage

Stores the toolpath setup name. Available in Mill, Turn and Mill/Turn.

TLP_SETUP_DESC

Type CHARACTER

Usage

Stores the toolpath setup description. Available in Mill, Turn and Mill/Turn.

TLP_FEAT_NAME

Type CHARACTER

Usage

Stores the toolpath feature name. Available in Mill, Turn and Mill/Turn.

TLP_FEAT_DESC

Type CHARACTER

Usage

Stores the toolpath feature description. Available in Mill, Turn and Mill/Turn.

TLP_OPER_NAME

Type CHARACTER

Usage

Stores the toolpath operation name. Available in Mill, Turn and Mill/Turn.

TLP_OPER_DESC

Type CHARACTER

Usage

Stores the toolpath operation description. Available in Mill, Turn and Mill/Turn.

New - Additional Variables**SETUP_ID****N_SETUP_ID****P_SETUP_ID***Type* INTEGER*Usage*

Stores current, next and previous setup ID number. Available in Mill, Turn and Mill/Turn.

TOOLPATH_ID**N_TOOLPATH_ID****P_TOOLPATH_ID***Type* INTEGER*Usage*

Stores current, next and previous toolpath ID number. Available in Mill, Turn and Mill/Turn.

NEXT_SETUP_4AX_ANGLE*Type* DECIMAL*Usage*

Stores the next 4th axis preposition setup angle. Available in Mill only.

NEXT_SETUP_5AX_ANGLE*Type* DECIMAL*Usage*

Stores the next 5th axis preposition setup angle. Available in Mill only.

PART_STOCK_DIAMETER*Type* DECIMAL*Usage*

Stores the stock diameter from the stock definition. Available in Turn and Mill/Turn.

PART_STOCK_LENGTH*Type* DECIMAL*Usage*

Stores the stock length from the stock definition. Available in Turn and Mill/Turn.

NEXT_OPR_TYPE*Type* INTEGER*Usage*

Stores the next operation type. Available in Mill, Turn and Mill/Turn.

NEXT_OPR_SUB_TYPE*Type* INTEGER*Usage*

Stores the next operation drill cycle type. Available in Mill and Mill/Turn.

NEXT_IS_LATHE*Type* INTEGER*Usage*

Stores if the next operation is lathe or not. Available in Mill/Turn.

NEXT_IS_LATHE=TRUE or FALSE

QUERY_ITEM_ID*Type* INTEGER*Usage*

Stores the ID of the object you are trying to get information from. Available in Mill, Turn and Mill/Turn.

QUERY_ITEM_ID=QUERY_INT_X_SETUP_DIR

QUERY_INT_X_SETUP_DIR holds the object ID of X Axis machining direction in the mill setup Axis tab.

Part Mode:

QUERY_INT_VAL= 1 Angle is selected.

QUERY_INT_VAL= 2 Automatic to stock is selected.

QUERY_INT_VAL= 3 Edge is selected.

QUERY_INT_VAL= 4 Sketch is selected.

Assembly Mode:

QUERY_INT_VAL= 1 Angle is selected.

QUERY_INT_VAL= 2 Automatic to stock is selected.

QUERY_INT_VAL= 3 Edge is selected.

QUERY_INT_VAL= 4 Sketch is selected.

QUERY_INT_VAL= 5 Fixture Coordinate System.

QUERY_RESULT

Type INTEGER

Usage

Stores the return value of the QUERY_SYSTEM() command. Available in Mill, Turn and Mill/Turn.

QUERY_RESULT=1 if successful.

QUERY_ERROR

Type INTEGER

Usage

Stores the error value of the QUERY_SYSTEM() command. Available in Mill, Turn and Mill/Turn.

QUERY_ERROR=0 for no error. If greater than zero, then it could be different types of errors currently not returned.

QUERY_INT_VALUE

Type INTEGER

Usage

Stores the integer value of the QUERY_SYSTEM() command. Available in Mill, Turn and Mill/Turn.

QUERY_INT_VALUE will equal any integer information retrieved from the QUERY_SYSTEM() command.

QUERY_DEC_VALUE

Type INTEGER

Usage

Stores the decimal value of the QUERY_SYSTEM() command. Available in Mill, Turn and Mill/Turn.

QUERY_DEC_VALUE will equal any decimal information retrieved from the QUERY_SYSTEM() command.

IS_5AXIS

Type INTEGER

Usage

IS_5AXIS = 0 or IS_5AXIS = FALSE

IS_5AXIS = 1 or IS_5AXIS = TRUE

If True, then it is either a Mill or Mill/Turn multiaxis operation.

IS_MILL_FACE

Type INTEGER

Usage

IS_MILL_FACE = 0 or IS_MILL_FACE = FALSE

IS_MILL_FACE = 1 or IS_MILL_FACE = TRUE

If True, then it is a Mill/Turn operation on the FACE.

IS_MILL_OD*Type* INTEGER*Usage*

IS_MILL_OD = 0 or IS_MILL_OD = FALSE

IS_MILL_OD = 1 or IS_MILL_OD = TRUE

If True, then it is a Mill/Turn operation on the OD.

IS_WRAPPED*Type* INTEGER*Usage*

IS_WRAPPED = 0 or IS_WRAPPED = FALSE

IS_WRAPPED = 1 or IS_WRAPPED = TRUE

If True, then it is a Mill operation on the OD.

New* - System Commands*QUERY_SYSTEM()***Type* New system command for all systems*Usage*

This command will query any object ID you pass to it. It can get information from an object that is in a Setup or operation dialog box. Available in Mill, Turn and Mill/Turn.

Associated commands and variables:

The example code below shows the other variables used with this command and that QUERY_SYSTEM() will first determine what has been selected for X Axis machining direction on the Axis tab in the Setup dialog box. If the value is "1" then "Angle" was selected and we can now do another QUERY_SYSTEM() to find the angle value that will be passed to QUERY_DEC_VALUE.

Note:

Currently QUERY_INT_X_SETUP_DIR and QUERY_DEC_X_SETUP_ANGLE are the only object ID's set. In the future you can send in an enhancement request to query other objects and we will add them when R&D resources are available.

```
:C: QUERY_ITEM_ID=QUERY_INT_X_SETUP_DIR
:C: QUERY_SYSTEM()
:C: IF QUERY_RESULT=1 THEN
:C:   F QUERY_INT_VAL=1 THEN
:C:   QUERY_ITEM_ID=QUERY_DEC_X_SETUP_ANGLE
:C:   QUERY_SYSTEM()
:C:   IF QUERY_RESULT=1 THEN
:C:     MY_OPER_SETUP_ANGLE=QUERY_DEC_VAL
:C:   CALL(TEST_QUERY)
:C:   ENDIF
:C: ENDIF
:C: ENDIF
```

KILLSYSFILE()*Type* New system command.*Usage*

This command will delete any file that has a path in the new variable called SYSFILENAME.

Syntax:

```
:C: SYSFILENAME="drive:\folder\filename"
:C: KILLSYSFILE()
```

