

Contents	
■ General Operation	1
■ Mill Features	5
■ 2.5 Axis Milling.....	5
■ 3 Axis Milling	7
■ Multiaxis Machining.....	7
■ Assembly Mode	8
■ Turning	8
■ Technology Database	10
■ Simulation	10
■ Posting / UPG	11
■ Wire EDM	11

■ General Operation

Updated Build Info File

- Purpose** Report errors that have been corrected in the current build of CAMWorks 2007.
- Implementation** On the Start|Programs menu, select CAMWorks2007, then select Resolved CPR's. You can also open the file CW2007BuildInfo.rtf in the \lang\xxxx folder (where xxxx is the language: e.g., \lang\english) inside the CAMWorks 2007 folder.

New Lock CAMWorks Data

- Purpose** Provide ability to freeze or lock CAMWorks data for features, operations or the entire part model so the data cannot be modified.
- Implementation** The Lock command has been added to the NC Manager, feature and operation shortcut menus. Multiple features and operations can be selected. Locked items are identified in a different color in the tree.
- When the NC Manager is locked, the entire model is locked and any changes cannot be saved. You can still manipulate the tree views, change parameters, insert features, etc.
 - Locked features cannot be renamed or edited. Any command that does not change the locked feature size or shape can be used (e.g., delete, reorder, hide, copy, suppress). If the Rebuild command is selected, a locked feature is not updated and toolpaths associated to the feature are not regenerated. During Rebuild, a new feature is created if the feature is different from the original.
 - For locked operations, any command that would modify the toolpath or operation parameters is disabled. Locked operations cannot be renamed. A locked operation can be post processed, simulated, copied, deleted, reordered, suppressed, hidden, etc.
- Locked items can be unlocked by selecting Unlock Node on the shortcut menu.

New Operation Recycle Bin

- Purpose** Provide a Recycle Bin for deleted operations.
- Implementation** A Recycle Bin has been added to the Operation tree. Only the operation and associated features links are stored with the deleted operation. The Recycle Bin shortcut menu includes commands to Restore All, Empty, Hide and Collapse Items. Operations in the Recycle Bin can be restored or removed using command on the operation shortcut menu.

Improved Feature Recycle Bin

- Purpose** Move all deleted features to the Recycle Bin.
- Implementation** When any feature created using AFR or IFR is deleted, CAMWorks moves the feature to the Recycle Bin. The features can be restored if necessary. In previous releases, only features found using AFR were stored in the Recycle Bin.

Improved Rebuild

- Purpose** Prevent IFR features created during AFR, such as Face, Open Pocket and taper/fillet features from being rebuilt if in the Recycle Bin.
- Implementation** Any feature in the Recycle Bin that was created using the options on the Features tab in the CAMWorks Options dialog box will not be restored to the Setup when Rebuild is run.

Improved Stock Outline, Setup Origin Display

- Purpose** Provide an option to display the stock outline continuously and automatically display the Setup origin.
- Implementation** The *Show stock outline* option is on the Display tab in the CAMWorks Options dialog box. When this option is selected, the stock displays continuously. When not selected, the stock displays only when the stock item is selected in the tree.
- The Setup origin displays automatically when a Feature or Operation is selected in the tree and during simulation.

Improved Saving Files With Large Amount of Toolpaths

- Purpose** Improve performance.
- Implementation** The time required to save files with a large amount of toolpaths has been reduced significantly. This is most noticeable in parts with 3 axis toolpaths.


New Display Main Parameters for Operations

- Purpose** Display main parameters for the selected operation in the tree.
- Implementation** The *Display tool tip for operation parameters* option has been added on the Display tab in the Options dialog box. When this option is checked and the cursor is placed over an operation in the tree, the main parameters for the operation display in the graphics area.

Improved Mill Spindle Speed

- Purpose** Provide an option to lock the spindle speed so that the RPM value will not update when the tool diameter changes.
- Implementation** On the F/S tab in the Operation Parameters dialog box, enable the *Lock spindle speed* option. The default for each operation can be defined in the Technology Database.

New Support for Multiple Machine Types in One File

- Purpose** In Part mode, allow multiple machine types to be supported in one file with support for SolidWorks configurations.
- Implementation** Multiple CAMWorks datasets are supported in Part mode. Each dataset is called a configuration. You can use configurations to support multiple machines and SolidWorks configurations.
-  When you double-click the Configurations item in the tree or right click and select Manage Configurations, the CAMWorks Configurations Manager dialog box displays. You can activate a different configuration, create new configurations, delete and copy configurations.
- Note: Multiple machine support will be implemented for Assembly mode in a future release.

Improved Hidden Feature and Operation Display

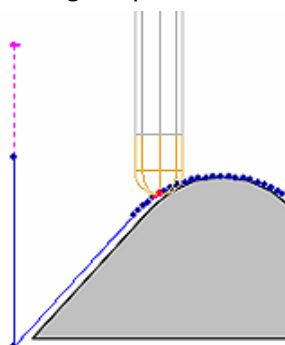
- Purpose** Display hidden features and operations in a different, user-definable color.
- Implementation** Hidden features and operation items display in a different color in the Feature and Operation trees. The color can be changed on the Display tab in the CAMWorks Options dialog box.

New Options for 4th and 5th Axis Indexing

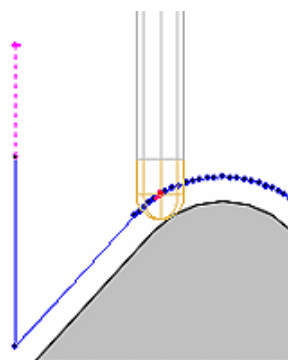
- Purpose** Provide the ability to define a Fixture Coordinate System in Part mode for 4th and 5th axis indexing.
- Implementation** The Fixture Coordinate System parameter has been enabled in Part mode on the Setup tab in the Machine dialog box. You can pick the Fixture Coordinate System and identify it rotates around an axis of the coordinate system. The axis and the 0 degree position on the 4th Axis and 5th Axis tabs can be based on the Fixture Coordinate System.

Improved G-code Output for Hog and Ball Nose tools

- Purpose** Allow G-code output for ball nose, hog nose, tapered ball nose and tapered hog nose tools to be output at the tool tip or the tool center.
- Implementation** For any 2 or 3 axis operation that supports ball and hog nose tools, the Mill Tool tab in the Operation Parameters dialog box contains an option to Output through Tip or Center.



Output through tip



Output through center

■ Mill Features

New Manufacturing View (MfgView) Feature Recognition

- Purpose** Provide a new generation of automatic feature recognition that finds additional feature types and allows features to be edited.
- Implementation** A Method option has been added on the Mill Features tab in the CAMWorks Options dialog box. When MfgView is selected, CAMWorks uses a new method to generate features. Non-hole features can be edited the same as interactively inserted features.

New Automatic Feature Recognition from Setup Level

- Purpose** Provide the ability to recognize Mill features from a user-defined direction only.
- Implementation** When the Method on the Mill Features tab is set to MfgView, you can right click a Mill Part Setup and select Recognize Features.

Improved Associate Any Mill Feature

- Purpose** Provide an option in the Associate Features dialog box to associate any Mill features to current features, not just features that do not rebuild.
- Implementation** The Associate Features command is always enabled on the NC Manager shortcut menu. An *All features* option has been added in the Associate Features dialog box. When this option is not checked, the dialog box lists only features that cannot be rebuilt. When checked, all existing features are listed.

New Support for Wrapped Features in Mill

- Purpose** Provide support for Wrapped features in Mill mode.
- Implementation** Wrapped features, previously supported only in Mill-Turn, can be inserted in Mill under the following conditions:
- A Rotary Milling license is required.
 - The current Mill Setup must be perpendicular to the 4th axis.
 - Indexing must be set to 4 Axis on the Setup tab in the Machine dialog box.
 - A Fixture Coordinate System has been enabled in Part mode and must be specified on the Setup tab in the Machine dialog box.
 - The X or Y axis of the Fixture Coordinate System must be selected to define the 4th axis.
 - Faces/edges used to define the Wrapped feature must be cylindrical.
 - The axis of the cylindrical faces/edges must be coincident with the 4th axis.

■ 2.5 Axis Milling

Updated Finish Mill Renamed Contour Mill

- Purpose** Change label for Finish Mill operation to more accurately identify capabilities.
- Implementation** The label for Finish Mill in the Operation tree has been changed to Contour Mill and the Finish tab in the Operation Parameters dialog box has been changed to Contour.

Improved Contour Mill Options for Multiple Cut Depths and Side Passes

- Purpose** Provide options to reduce and eliminate unnecessary moves when contour (finish) machining features with multiple depths and side passes.
- Implementation**
- When machining features with multiple cut depths and/or side passes, it may be more efficient to reverse the cut direction from cut depth to cut depth and/or from side pass to side pass.
A Zigzag option has been added in the Depth processing section on the Contour tab and in the Side Parameters dialog box. On the Contour tab when Depth processing is set for *To depth by region* and this option is checked, the tool moves down in Z without retracting and machines in the opposite direction. When checked in the Side Parameters dialog box, the tool reverses direction from side pass to side pass.
 - Cut depth and Side passes parameters have been added to the Leadin tab to specify how to link one depth to the next and one side pass to the next: Retract to Clearance, Stay Down or Direct (for side passes). When Stay Down is selected, the Feedrate for the feed move that joins the current leadout to the next leadin can be defined as a value or as a percent of the XY feedrate value on the F/S tab.

Improved Contour Mill Compensated Toolpath Output

- Purpose** Provide compensated toolpath output for Contour Mill operations when the toolpath center is set to Without compensation.
- Implementation** Done automatically for Contour Mill operations when the NC parameters are set to:
CNC compensation = On
Toolpath center = Without compensation
- In previous releases, the CL that was output to posting is not correct for some cases and other cases the machine tool would misinterpret the output and generate incorrect moves on the machine. The method for the toolpath that was output to the CL file was to effectively convert the feature geometry to CL records. In some cases this output was processed properly by the machine tool, but in some cases it was not and the machine tool would incorrectly cut the part.
- CAMWorks can now output CL data to correctly cut the part for the following cases:
- When the corner radii are smaller than the sum of the tool radius and XY allowance many machines will not generate the proper toolpath.
 - When islands and/or the feature perimeter would merge into a single shape when machined or when the shape that results would be two areas instead of one.
 - Where the XY allowance is negative and for tapered and filleted features.

■ 3 Axis Milling

Updated Labels for Advanced 3 Axis Operations Simplified

Purpose	Change labels for Advanced 3 Axis operations to simplify names and more accurately identify capabilities.
Implementation	The labels for Adv. 3 Axis operations have been changed as follows: Adv. 3 Axis Rough -> Area Clearance Adv. 3 Axis Z Level -> Z Level Adv. 3 Axis Pattern Project -> Pattern Project Adv. 3 Axis Stepper -> Constant Stepper Adv. 3 Axis Curve Project -> Curve Project Adv. 3 Axis Flat Area -> Flat Area

Improved Arc fitting for all 3 Axis Mill operations

Purpose	Provide the ability to generate toolpaths comprised of line and arc moves for all 3 Axis Mill operations.
Implementation	Arc fitting options are on the Advanced tab in the Operation Parameters dialog box for all 3 Axis operations (except 3 Axis Rough and 3 Axis Finish). Previous releases supported arc fitting for Area Clearance, Z Level and Flat Area only. Using Arc fitting can be beneficial even when the machine does not support circular interpolation and your post does not support it. When arc fitting is used, less memory is required for the display, the CAMWorks file size is smaller and simulation is faster especially in Tool mode. In benchmark testing on some parts, the CAMWorks toolpath size was almost 6 times smaller and the simulation was almost 4 times faster.

■ Multiaxis Machining

Improved Spiral pattern cutting

Purpose	Provide the ability to generate spiral toolpaths for all patterns.
Implementation	The <i>Change parallel cuts to spiral</i> option on the Advanced tab previously supported only the Slice pattern. This option can now be selected for all pattern types on the Pattern tab.

New Side Tilt Strategy when tool axis is tilted relative to the cutting direction

Purpose	Provide an additional side tilt strategy when the tool axis is tilted relative to the cutting direction.
Implementation	On the Axis Control tab, when using Tilted Relative to Cutting Direction for the tool axis cutting direction tilt, the option <i>Use Tilt Line Definition</i> has been added to the list of options for the Side tilt strategy. When this option is selected, you can select lines to define the side tilting direction.

New Line Tilt Strategy when tool axis is tilted through lines

Purpose	Provide an additional line tilt strategy when the tool axis is tilted through lines.
Implementation	On the Axis Control tab, when using Tilted Through Lines for the tool axis cutting direction tilt, the option <i>Always Closest Two Lines</i> is now supported for the Line tilt strategy. When this option is selected, CAMWorks approximates the direction of the tool axis using the two closest lines. The result is that the tool axis will have the same orientation as the tilt axis.

New Roughing options

Purpose	Provide an additional options on the Roughing tab.
Implementation	New options have been added to the Roughing tab: <i>Connect by shortest distance</i> When this option is checked, CAMWorks uses the shortest distance to the next cut. <i>Tangent ramping</i> The option is enabled when Axial cut depths is selected. When this option is checked, CAMWorks changes the single slices to one spiral slice. The order is starting from the roughing passes to the finishing passes. The tool starts and stops on the same position.

■ Assembly Mode

New Machining of SolidWorks Part Configurations in Assembly Mode

Purpose	In the Part Manager, identify parts based on the combination of the part name and the configuration name so that multiple configurations of a part can be machined at the same time.
Implementation	In the Part Manager, when a part model is selected, it will be identified in CAMWorks as well as Part Manager with both the part name and part configuration. Multiple instances of the same part name/configuration name will still be supported. When AFR is run, it will be run for each different combination of part name/configuration name.

New Split Instances of Same Part

Purpose	Provide ability to split instances of the same part and generate separate features and operations for each instance.
Implementation	In the Part Manager, when a part model is selected, click the Split Instances button. When AFR is run, separate instances of each feature are generated. Separate operations are also generated. This option can be used when more flexibility is required (e.g., when sequencing operations or machining features differently).

Improved Assembly Mode Feature Names

Purpose	Simplify identifying a feature in the Feature Manager with the corresponding feature in a Setup.
Implementation	If a feature is renamed in the Feature Manager, the feature name in the Setup is changed automatically.

■ Turning

New Associate Turn Features

Purpose	Provide an option to associate new Turn features to current Turn features.
Implementation	Select Associate Features on the NC Manager shortcut menu. In the Associate Features dialog box, check the <i>All features option</i> . Highlight a Current feature and click Associate. Select Add to generate operations for the new feature that are copies of the operations generated for the selected Current feature. Select Replace to replace the selected Current feature with the new feature in every operation that contains the Current feature and delete Current feature from the Feature tree.

Improved Turn Cutoff Chamfer Extension

- Purpose** Provide the ability to specify a chamfer extension distance to increase the width of the toolpath for machining the chamfer.
- Implementation** A new *Chamfer ext.* option on the Cutoff tab in the Operation Parameters dialog box allows you to define a distance above the diameter of the cutoff feature where the chamfer will start. You can use this option to eliminate small bumps that may occur at the top of the chamfer due to tool wear or the diameter of the stock not being exactly the same as the feature diameter.

Improved Turn Cutoff Slowdown Option

- Purpose** Provide the ability to decrease the speed and feedrate at a user specified length from the end of a cutoff toolpath.
- Implementation** Options have been added to the Cutoff tab in the Operation dialog box to reduce the spindle speed and feedrate at a specified distance from the end of the Cutoff feature.

New Rough Groove Cleanup Pass

- Purpose** Provide an option to generate a cleanup pass in rough grooving to remove scallops generated by the rough groove toolpath.
- Implementation** The following options on the Rough Groove tab allow you to define whether to generate a cleanup pass and what method to use:
- Cleanup pass*
- None - No cleanup pass is generated.
 - From WIP - Cleanup pass is generated based on the WIP model.
 - Full Groove - Cleanup pass is generated for the entire groove shape.
- Previous allowance* -The perpendicular distance between the original (uncut) and final surfaces of the stock.
- Cut amount* - Defines how deep the cut is. If the Previous allowance is larger than the Cut amount, more than one pass is generated.

New Finish Groove Cutter Compensation

- Purpose** Support cutter compensation for Finish Groove toolpaths.
- Implementation**
- In order to generate the leadins and leadouts needed for machine compensation, the Leadin/Out tab has been added to the Finish Groove operation.
 - On the Finish Groove tab, an option allows you to specify whether to output compensation codes in the NC program.

New Groove Tool Driving Point

- Purpose** When using a groove tool for a turn toolpath, provide options to define the driving point of the groove tool.
- Implementation** CAMWorks computes and displays the toolpath as if the driving point is at the center of the insert. However, in many cases the XZ point coordinates in the G-code output must be at the edge of the insert. CAMWorks provides several options for controlling the output driving point on the groove tool. A Driving point option has been added for all operation types that support groove tools (e.g., Rough Groove, Finish Groove, Cutoff, Rough Turn, Finish Turn).

Improved Turn Thread End Length Parameter

- Purpose** Provide an end length parameter to increase the length of the Thread toolpath.
- Implementation** The End length parameter on the Thread tab in the Operation Parameters dialog box defines an incremental amount to add to or subtract from the end of the feature length and the resulting toolpath.

Improved Turn Thread Chamfer Angle Parameter

- Purpose** Allow a user-defined Chamfer angle.
- Implementation** New options on the Thread tab in the Operation Parameters dialog box allow you to specify a chamfer with a user-defined angle.

New Set Minimum and Maximum Z Limits for Turn Operations

- Purpose** Provide the ability to set an absolute machine depth for Turn operations.
- Implementation** User-defined Z limits can be set globally on the Advanced tab in the Part Setup Parameters dialog box or for each operation on the Advanced tab in the Operation Parameters dialog box.

■ Technology Database

New TechDB Support for Wire EDM


- Purpose** Provide TechDB support for Wire EDM features and operations.
- Implementation** An EDM section has been added to the TechDB that allows you to define user-defined attributes, EDM stock materials, and EDM features and operations. Note that EDM Machines are not defined in the TechDB. They are defined in CAMWorks as in previous releases.

New Set Defaults for New Operation Parameters

- Purpose** Provide options to set defaults for operation parameters added in CAMWorks 2007 for all modules.
- Implementation** The default values for operation parameters added in this release can be defined on the applicable forms in the TechDB.


■ Simulation

New Reverse Toolpath in Tool Mode

- Purpose** Provide a simulation control to reverse the toolpath while in Tool mode.
- Implementation**  During simulation, a Reverse step button is enabled on the Toolpath Simulation toolbar for Mill, Turn, Mill/Turn and Wire EDM. When selected, the tool moves backward one CL record for each time the button is clicked. The Simulation tab in the CAMWorks Options dialog box includes an option to specify the maximum number of cuts where material can be added to the stock while in Reverse step mode. Note that when using this option, Tool mode simulation will be slower by about 15%. As the number of allowed steps to add material is increased, more system memory is also required. A large number could result in an out of memory condition and further slow down simulation.

New Section View of Simulated Mill or Wire EDM Part

Purpose Provide functionality to create a section view of a simulated Mill or EDM part.

Implementation  A Section view button has been added to the Toolpath Simulation toolbar for Mill and Wire EDM. When the Section View button is clicked, you can customize the section view by selecting a CAMWorks plane that the section plane will be parallel to and picking an entity to define the section plane.

Improved Simplified Turn Thread Simulation Option

Purpose Provide an option to simulate a turn thread in a simplified form to improve performance.

Implementation The Simplified threading option on the Simulation tab in the CAMWorks Options dialog box can be used to simulate a thread in a simplified form to improve performance. When this option is not checked, CAMWorks simulates the full thread, which can result in very slow simulation speeds. When this option is checked, CAMWorks simulates the thread as if the cut is a normal turning cut and thread simulation should be much faster.

Improved Setup Origin Display During Simulation

Purpose For Mill and Wire EDM, provide an option to automatically display the Setup origin during simulation.

Implementation Done automatically.

■ Posting / UPG

New CAMWorks Turn System Variables

Purpose Provide additional system variables for CAMWorks Turn.

Implementation Support for new variables has been added for CAMWorks 2007 Turn. Post customization is required in order to use these variables.
For more information, install the UPG from the CAMWorks 2007 CD, start the UPG and select Complete Reference Guide on the Help menu. This information is also in the Post Processor Writer's Reference on the Miscellaneous Files downloads page on the TekSoft website.

■ Wire EDM

New TechDB Support

Purpose Provide TechDB support for Wire EDM features and operations.

Implementation An EDM section has been added to the TechDB that allows you to define user-defined attributes, EDM stock materials, and EDM features and operations. Note that EDM Machines are not defined in the TechDB. They are defined in CAMWorks as in previous releases.

New Feature Attributes

Purpose Provide support for feature attributes.

Implementation Assigning an attribute to an EDM feature is similar to assigning Mill feature attributes. This includes automatic assignment for AFR features and assignment of attributes in the Insert 2.5 Axis EDM Feature and Insert 4 Axis EDM Feature dialog boxes. A new Feature Parameters dialog box allows you to change the attribute for the selected feature.

New Associate EDM Features

- Purpose** Provide an option to associate new EDM features to current EDM features.
- Implementation** Select Associate Features on the NC Manager shortcut menu. In the Associate Features dialog box, check the *All features option*. Highlight a Current feature and click Associate. Select Add to add the new feature to all operations for the selected Current feature. Select Replace to replace the selected Current feature with the new feature in every operation that contains the Current feature and delete Current feature from the Feature tree.

New Separate Leadin/Leadout for Rough and Skim Cuts

- Purpose** Allow separate leadin/out methods for the EDM Rough/Glue Stop passes and for the Skim/Glue Stop passes.
- Implementation** For 2 Axis and 4 Axis Contour operations, options have been added on the Leadin/Out tab that allow you to apply the same leadin/leadout method to rough and skim passes or apply a separate leadin/leadout to rough and skim passes.

Improved Output Options

- Purpose** Provide the ability to output rough cuts only, tab cuts only, or rough and tab cuts.
- Implementation** For 2 Axis and 4 Axis Contour operations, an Output option on the EDM tab allows you to select Rough cut and/or Tab cut.

New Automatic Punch and Die

- Purpose** For 2 Axis Contour operations, provide an option to automatically generate toolpath passes that will cut both a die and punch at the same time.
- Implementation** For 2 Axis Contour operations that are machining a 2 Axis die feature with no taper, the *One Cut Punch & Die* option can be selected in the Land and taper section on the EDM tab.

New Option for G-code to Output an Offset Toolpath

- Purpose** Provide the ability for the G-code to output to the wire center.
- Implementation** In previous releases, except for coreless toolpaths, all G-code output for 2 axis and 4 axis toolpaths was to the feature geometry, which required that all compensation be performed on the machine tool control. For various reasons, it is sometimes required that the G-code output XY values to the wire center and per offset pass.
- The Posting tab in the 2 Axis and 4 Axis Contour dialog boxes has a Toolpath center option that defines whether the G-code output should be to the wire center (With compensation) or the feature geometry (Without compensation).