



What's New in

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Supported Platforms

Supported Platforms for 64-bit		
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2022 - SOLIDWORKS 2021 - SOLIDWORKS 2020 - CAMWorks Solids 2022 - CAMWorks Solids 2021 - CAMWorks Solids 2020	
Operating System:	 64-bit version of: Windows 10 Windows 8.1 Windows 7 (SP1 or higher) [*Home Editions are not supported] Note: CAMWorks 2021 is supported only on 64-bit Operating systems.	

Purpose:	The Resolved CPR (<i>CAMWorks Problem Report</i>) document has been updated to report the software errors that have been resolved in the current Service Pack (SP5).
Implementation:	To view the document, select: Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.



License

New - License Activation Method: Online Activation using Activation IDs

Purpose:

The **Online Activation** method (introduced from **CAMWorks 2021 SP5** version onwards) employs 'Activation IDs' that functions as software keys for activating the products and modules within the **CAMWorks** suite of application.

Implementation:

These Activation IDs for products in the CAMWorks suite are alphanumeric in nature and consist of 32 characters. When renewing/purchasing the CAMWorks subscription, you need to specify your license type (Nodelocked/Floating Network), the desired CAMWorks modules/products you wish to use and then pay the applicable license fees. HCL Technologies Ltd. (IP owners of the CAMWorks software application) will then provide you with one or more Activation IDs for activating the purchased products.

Once received, these Activation ID(s) must be input by you in the relevant user interface of the **CAMWorks License Manager** tool for license activation. These Activation ID(s) are then validated and verified online. If successfully verified, then the CAMWorks application (with specific products and modules paid for by you) will be activated.

Status of Previously Existing License Activation Method of 'License File Based Activation'

Online Activation method for license activation has been newly introduced in *CAMWorks 2021 SP5* version. The previously existing **License file based** method for license activation will continue to remain available within the **CAMWorks License Manager** tool. However, do note that the license file based method for license activation will gradually be phased out.

As of now, in the **CAMWorks 2021** version, you will have the option to choose either the **Online Activation** or **License file based** method of license activation when purchasing/renewing your *CAMWorks* subscription.

We recommend that you switch to the more convenient **Online Activation** method.

	🖗 CAMWorks License Manager	×
	Status License Setup Authorized Modules Server Settings	
	License method : Nodelocked (Standalone)	
	(Supports CAMWorks 2022 and newer) Configure FlexLM license Use this option to activate your license based on the Activate License Last license activated on : Not known	
Nodelocked	License: Separate Tabs in License Setup Tab for Online Activation and Based Activation	I Licer

🖗 CAMWorks License Manager	×
Status License Setup Authorized Modules S	erver Settings
Server License Method : Online Activ	ations 💽 🤅
Server license settings Online Activ	ations
<u>A</u> ctivate License	on the Activation ID(s) sent to you. Last license activated on : Not known
Show Licenses	Use this option to view activated licenses.
<u>R</u> eactivate Existing	Use this option if you want to update/upgrade your licenses or if you are facing any issues with your licenses.
Deactivate License	Use this option to release license from this machine.
Options	
Server Port(Mandatory) : 27000	Secondary Port(Optional) :
Start License Service	Use command to install and start the license service.

Floating Network Server License: Separate Options provided for License Activation in Server Settings Tab

Activation Options For 'Online Activation' Method of License Activation

Depending on whether you have or do not have access to the Internet on the 64-bit Windows system (on which the CAMWorks license will be activated), following two options will be available for Online Activation

Automatic Method:

In *Automatic* method of *Online license* activation, a live Internet connection is required for activating the license. You need to input the Activation IDs provided to you and then have them validated. Once successfully validated, the activated *CAMWorks* products will be displayed within the *CAMWorks Product Activation - Currently Activated Products* dialog box.

Manual Method:

This option is recommended only if you do not have a live internet connection. In this method of *Online License* activation, you need to create a license request file using the Activation IDs (Entitlement IDs) provided to you and then email it to register@camworks.com. As a response, you will receive a response file attachment via email. You need to load this response file to activate your CAMWorks products. Once successfully validated, the activated CAMWorks products will be displayed within the *CAMWorks - Currently Activated Products* dialog box.

Reactivating your License using Online Activation Method

If you wish to upgrade or renew your CAMWorks license, the **Online Activation** method provides separate **CAMWorks Product Reactivation** user interfaces within the CAMWorks License Manager tool to reactivate your license using your existing Activation IDs. (Both **Automatic** and **Manual** options of license reactivation are available.)



CAMWorks Product Reactivation × Reactivate Your CAMWorks Product Selected products listed below will be reactivated. How would you like to reactivate? Automatically over the Internet (Recommended) Manually via E-Mail (Add all activation IDs and then click on activate) Select all 🖂 Please select the product that need to be reactivated : Product Expiry Date USP Date Activation ID Status Commercial Permanent 31-Oct-2022 2420-4345-38b5-43f5-a717-023a-3663-e514 Active CAMWorks Milling Professional 31-Oct-2022 Permanent 2420-4345-38b5-43f5-a717-023a-3663-e514 Active CAMWorks Solids P Permanent 31-Oct-2022 2420-4345-38b5-43f5-a717-023a-3663-e514 Active Commercial Permanent 31-Oct-2022 2420-4345-38b5-43f5-a717-023a-3663-e514 Active ✓ I have read the HCL Technologies Privacy Policy and agree to its terms and conditions. Reactivate My Product(s) Close Help

User Interface for Reactivating your License via Online Activation Method

Advantages of 'Online Activation' Method of License Activation – Prompt Deactivation and Activation

In the previously used *License file based* method, users had to inform support about the intention to switch system, raise a new license request file from the new system, then wait for one business day to get the license file required for product activation. Consequently, shifting a *Nodelocked* or *Floating Network* installation from one system to another was a bit cumbersome. This issue is addressed with the *Automatic* option of *Online Activation*. To switch from one system to another, use the *CAMWorks Product Deactivation* UIs available within the *CAMWorks License Manager* tool to deactivate your products from the current system and then activate it on another system (using the *CAMWorks Product Activation* UIs). This process requires no intervention from CAMWorks Support or waiting for any email replies. (Both *Automatic* and *Manual* options of license deactivation are available.)





	🐔 CAMWorks License Manager		×
	Status License Setup Authorized Modules Server	Settings 1. Click Serv	ver Settings
	Server License Method : Online Activations		• •
	Server license settings	nline Activations' Op	ption
	Activate License Us	e this option to activate your lic the Activation ID(s) sent to you st license activated on . Not kn	ense based I. own
	3. Click on 'Activate License' bu	utton	
eps 1-3:	Click 'Activate License' Button in Server	Settings Tab for Floati	ng Network Server
	License		
	See CAMWorks Product Activation		×
	License Activation To activate your CAMWorks product you must use	the Activation IDs.	
	How would you like to activate?		
	Automatically over the Internet (Recommended)	4. Select this opt	tion.
	O Manually via E-Mail (Add all activation IDs and	then click on activate)	
	Enter your Activation ID(s) and click on add : 5. Inpu	t Activation ID in b	elow field.
			Add
	6f94-e950-47fd-4b44-9c6e-acaa-e435-7675	6. Click A	dd button.
	Find My Product(s		
	Products associated with above IDs :	7. Click Find My	Product(s).
	Product	ExpiryDate	USPDate
	CAMWorks Milling Professional	Permanent	31-Oct-2022
	CAMWorks Solids P	Permanent	31-Oct-2022
	Commercial	Permanent	31-Oct-2022
	8. Place a check in below checkbo	X. <u>Click here to open t</u> and agree to its terms and co	the license portal nditions.
	9. Click 'Activate My	Product(s) button.	
	Activate My Product	(s) Close	Help
tep 4-9:	In the next UI that is displayed, select 'Aut	omatic' option, input A	Activation ID(s), Agree



	Name* :	XYZ		
	Email-ID* :	XYZ@mail.com		
	ОК	Close	Help	
Ste	p 10: In the	next UI, input user de	tails and click OK	
<u> </u>				
줄 CAN	Works Produc	t Activation - Currently Act	ivated Products.	×
Seculto	Works Produc	t Activation - Currently Act	ivated Products.	×
Se CAN	IWorks Produc	t Activation - Currently Act	ivated Products.	×
Se CAN Results	Works Produc	t Activation - Currently Act	ivated Products.	×
Results	Works Produc	t Activation - Currently Act	ivated Products. ated products.	×
Se CAN Results	Works Produc	et Activation - Currently Act	ivated Products. ated products.	×
CAN Results	Works Produc	et Activation - Currently Act below shows currently active	ivated Products. ated products.	×
CAN Results	Works Produc	t Activation - Currently Act below shows currently activ fucts Expiry Date	ated products. ated products. USP Date	×
CAN Results	Works Produc	et Activation - Currently Act below shows currently active fucts Expiry Date Permanent	ated products. USP Date 31-Oct-2022	×
Currently Product	Works Produc The list I Activated Proc	t Activation - Currently Activation shows currently activation below shows currently activation below shows currently activation below shows currently activation below	ated products. USP Date 31-Oct-2022 31-Oct-2022	×
Currently Commer CAMWO	Works Produc The list I Activated Proc	tt Activation - Currently Act below shows currently activ ducts Expiry Date Permanent fess Permanent fers Permanent	ated products. USP Date 31-Oct-2022 31-Oct-2022 31-Oct-2022	

Step 11: In the next UI, the products successfully activated will be displayed. Click Close to exit.

Steps to Activate Your CAMWorks Products using 'Manual' Option of Online Activation

Refer the following steps accompanied by user interface images for details on how to activate license using *Manual* option of *Online Activation*. **Note:**

Steps 1 to 3 for this option are identical to Steps 1-3 for 'Automatic' Method.

🖗 CAMWorks License Manager 🛛 🗙	
Status License Setup Authorized Modules Server Settings 1. Click Server Settings	
Server License Method : Online Activations	
Server license settings	
Activate License on the Activation ID(s) sent to you. Last license activated on : Not known	
3. Click on 'Activate License' button	
eps 1-3: Click 'Activate License' Button in Server Settings Tab for Floating Network License	(Serv

6	Wh	at's New in CAMWorks 2021 – SP5
	See CAMWorks Product Activation	×
	License Activation To activate your CAMWorks product you must use the Activ	vation IDs.
	How would you like to activate?	
	 Automatically over the Internet (Recommended) Manually via E-Mail (Add all activation IDs and then click 	4. Select this option. k on activate)
	5. Input Acti Enter your Activation ID(s) and click on add :	vation ID in below field.
		Add
	X000K-X000K-X000K-X000K-X000K-X000K-X000K	6. Click 'Add' button.
	 7. Place a check in below checkbox. ✓ I have read the HCL Technologies Privacy Policy and agree 	ee to its terms and conditions.
	Activate My Product(s)	Close Help
	8. Click 'Activate N	ly Product(s) button.
 CAMWorks Prod 1) Activation ID(s) v xxxx-xxxx-xxxx-xxxx 	duct Activation × which needs to be activated or deactivated	- -
2) Press save to cre Name* :	ate a request file 9. Input User Details below.	
Email-ID*: xv	- z@abc.com	Step 9-13: In the next UI that
Save	10. Click Save to generate Request file. Email Request file to HCL.	IS displayed, input user details, save and email the request file. Use the 'Open' button to browse and select the response file you receive
address to get a res	sponse file : (Login is required) camworks.com Click here to open portal in	Activate this file by clicking the 'Activate My Products'
4) Press open to lo	ad the response file 11. Download the response file	button.
Open	you receive. 12. Click Open to select and activate	
If you have received then proceed with rerun the Wizard Ia	the Response file. I the response file, load it by clicking the Open button and the below option. To discontinue, click Close. You can ter.	
Help 1	3. Click 'Activate My Product(s) button. Activate My Product(s) Close	



Supported Platforms

Supported Platforms for 64-bit		
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2022 - SOLIDWORKS 2021 - SOLIDWORKS 2020 - CAMWorks Solids 2021 - CAMWorks Solids 2020 Note: CAMWorks Solids 2022 will be supported once released.	
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] Note: CAMWorks 2021 is supported only on 64-bit Operating systems.	

Purpose:	The Resolved CPR (<i>CAMWorks Problem Report</i>) document has been updated to report the software errors that have been resolved in the current Service Pack (SP4).
Implementation:	To view the document, select: Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.



Supported Platforms

Supported Platforms for 64-bit	
	The 64-bit version of: - SOLIDWORKS 2021
Solid Modeler:	- SOLIDWORKS 2020
	- CAMWorks Solids 2021
	- CAMWorks Solids 2020
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] Note: CAMWorks 2021 is supported only on 64-bit Operating systems.

Purpose:	The Resolved CPR (<i>CAMWorks Problem Report</i>) document has been updated to report the software errors that have been resolved in the current Service Pack (SP3).
Implementation:	To view the document, select: Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.



Supported Platforms

Supported Platform	upported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2021 - SOLIDWORKS 2020 - CAMWorks Solids 2021 - CAMWorks Solids 2020	
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] Note: CAMWorks 2021 is supported only on 64-bit Operating systems.	

Purpose:	The Resolved CPR (<i>CAMWorks Problem Report</i>) document has been updated to report the software errors that have been resolved in the current Service Pack (SP2).
Implementation:	To view the document, select: Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.



Supported Platforms

Supported Platforms for 64-bit	
Solid Modeler:	The 64-bit version of: - SOLIDWORKS 2021 - SOLIDWORKS 2020 - CAMWorks Solids 2021 - CAMWorks Solids 2020
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] Note: CAMWorks 2021 is supported only on 64-bit Operating systems.

Resolved CPRs document

Purpose:	The Resolved CPR (<i>CAMWorks Problem Report</i>) document has been updated to report the software errors that have been resolved in the current Service Pack (SP1).
Implementation:	To view the document, select: Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.

New - Tutorial Document on Assembly Mode of Turn/Mill-Turn Module

Purpose:

A tutorial document illustrating how the Assembly Mode of Turn/Mill-turn module of CAMWorks can be used for machining assemblies

Implementation:

<u>Assembly mode support Turn/Mill-Turn module</u> was introduced in *CAMWorks 2021 SP0* version. A tutorial document with illustrative examples on this Turn/Mill-Turn Assembly mode will now be available from *CAMWorks 2021 SP1* version onwards.

Once **CAMWorks** is loaded as an Add-in within SOLIDWORKS/ CAMWorks Solids, this tutorial document can be accessed from the SOLIDWORKS/CAMWorks Solids **Help** menu by selecting **CAMWorks 2021>>Tutorials**. Executing this command will display a window listing all the CAMWorks documents in PDF format. Double-click on the document named **Assembly_Mode_of_Turn_MillTurn_Tutorial.pdf**. This document contains two tutorials illustrative examples of how to use the Assembly mode of the Turn/Mill-Turn module of CAMWorks.



Supported Platforms

Supported Platform	forms for 64-bit	
Solid Modeler:	The 64-bit version of: SOLIDWORKS 2021 SOLIDWORKS 2020 CAMWorks Solids 2020 Note: CAMWorks Solids 2021 will be supported when released.	
Operating System:	64-bit version of: - Windows 10 - Windows 8.1 - Windows 7 (SP1 or higher) [*Home Editions are not supported] Note: CAMWorks 2021 is supported only on 64-bit Operating systems.	

Purpose:	The Resolved CPR (<i>CAMWorks Problem Report</i>) document has been updated to report the software errors that have been resolved in the current Service Pack (SP0).
Implementation:	To view the document, select: Start>>All Programs>>CAMWorks2021x64>>Resolved CPR's.



Mill

New - Prompt for Rebuild of CAM data when Stock is Modified

Purpose:

Option to rebuild features, operations, and toolpath data whenever there is a change to the stock parameters

Implementation:

For Mill and Turn part models, many features and operations parameters can be associated with the assigned stock parameters. Any changes to the stock can affect these parameters and consequently, the generated toolpaths.

In the Turn module of CAMWorks, whenever the stock associated with a Turn part model is modified, a warning message that prompts users to regenerate the toolpaths is displayed.

From *CAMWORKS 2021* version onwards, this functionality has been extended to the Mill and Mill-Turn module too. In a Mill part, whenever the stock parameters are modified after the generation of features, operations, and toolpaths, then a warning message prompting user to rebuild the data will be displayed. The warning message box will list all the *CAMWORKS* entities in the part/assembly that will be affected by the change in stock. Such entities include Coordinate Systems, Mill Part Setups, Features and Operations. (These items are listed in the *CAMWORKS* Feature tree and Operation tree.)

- Click 'Yes' within this warning message box to rebuild the data.
- Click 'No' within this warning message box to ignore the warning. Data will not be rebuilt.
- If you do not wish to view this warning every time you change the stock parameters, place a check in the *Do not show this dialog* checkbox option within this warning message.

	Stock parameters have changed. The for to be rebuilt.	ollowing CAMWorks items need
	Do you want to rebuild the CAMWorks if	tems?
Coordina	te System (0)	^
Mill Part	Setups (1)	
Mill Part Se	tup1	
Features	(10)	
Open Pod	et1	
Face Feat	.re1	
Kola 1		>
Do not s	now this dialog	

Impact when you select 'No' within the Message Box

If you select 'No', the *CAMWORKS* entities in the Feature tree and Operation tree affected by the change in stock parameters will be flagged with a yellow exclamation mark appearing over their respective icons.



•

•





	Errors	×	
	Stock parameters have been modified. The toolpath need Generating the toolpath will clear the status of the opera	ds to be regenerated. tion node.	
	Generate toolpath	Close	
Error r	nessage displayed when 'What's Wrong' option in Context Node is selected	Menu for an Flagged Ope	eration
Controlli	ng the Display of the Warning Message		
The displa checkbox (<i>Options</i> By defaul <i>dialog</i> che (<i>Mill</i>) cheo message checkbox	ay of this warning message is controlled by the On Stock P coption in the Prompt to rebuild group box on the Updat command to display this dialog box is available on the <i>CAI</i> it, this checkbox option will be checked. When you place a eckbox within the Warning Message box, the status of the ckbox option will change to unchecked state. You can resu at any point of time by placing a check back in the On Sto contion	Parameters Modification (M te tab of the Options dialo MWorks Command Manag check in the Do not show On Stock Parameters Mod me display of the warnin ck Parameters Modificatio	ill) g box. er.) / this lification g n (Mill)
Opti	ions		×
Ger	neral Mill Features Display Simulation Update File Location	S	
		Reset All	
	On Rebuild		
	Add new features to Feature Tree (otherwise they are added in	n recycle bin)	
	Delete rebuild failed features without operations		
	Delete rebuild failed features with operations		
	Delete rebuild failed child features from group features		
	Generate operations for new features		
	Generate toolpaths for operations without toolpaths		
	Prompt to rebuild		
	✓ On file open		
	On CAMWorks tree selection and file save		
	On stock parameters modification (Mill)		
	View system defaults Apply to	All Possible V	
	ОК	Cancel Help	
	Option in Updates Tab of Options Dialog	g Box	



New - Option for Defining Cylindrical Stock Type in Mill Mode

Purpose:

To provide a functionality whereby users can define Cylindrical stock for Mill Parts

Implementation:

In previous versions of *CAMWORKS*, the stock type for a Mill part model/assembly could be defined from a bounding box, extruded sketch, STL file or another part file. The option to directly define a cylindrical stock type was not available.

From *CAMWORKS* 2021 version onwards, the option to define a stock of cylindrical shape will be available in the *Stock Manager* dialog box user interface for both Part mode and Assembly mode.

When the Cylindrical Stock type option is selected, users will have the following parameters to define the stock:

- Centre axis of the stock (Default Centre axis will be the Z axis of the assigned Fixture Coordinate System)
- Origin of the stock (Default origin will be the centroid of the mill part model)
- Diameter of Cylinder (Default diameter will be the minimum diameter based on the corresponding dimension of the part model such the part fits within it; refer image below.)
- Length of cylinder (Default length will be the minimum length based on the corresponding dimension of the part model such the part fits within it; refer illustrative image below.)
- Offsets, if any.



Mill Part Model with Cylindrical Stock of Minimum Diameter and Length Note that the Z Axis of Cylindrical Stock parallel to the Mill Part Setup

If the axis of the cylindrical stock is parallel to the Z axis of a Mill Part Setup for the part model, then you can define the origin of the Mill Part Setup using one of the three central vertices (Top Center, Mid Center and Bottom Center) of the cylindrical stock. This option is available in the **Origin** tab of the **Part Setup Parameters** dialog box.



Options to choose Origin of Mill Part Setup in Origin tab of Part Setup Parameters Dialog Box

Material : 1018	Cylindrical Stock
Low Carbon Alloy Steel	
Stock Type 🔅	Existing control to select Coordinate System
Co-ordinate System	
J. Origin	Ontion to colorat the suis
Axis: Z	for aligning the stock
Stock Locate	
7.9072in	1
0.375in	
0in 🔹	1
Minimum Dia and Length	
Offset Parameters :	
0in 📫	Cylindrical Stock Parameters
inin Oin	
u Oin ↓	
Load default Set as default	
Stock size A	
X: 7.9072in Y: 7.9072in Z:0.375in	
Number of Stocks $\&$ Parts :	
CAMWorks NC Manager	
Stock :	
Stock [Round Bar Stock]	



Improved - Option to Define Bounding Box Stock in Mill Mode using Pre-Defined Blocks

Purpose:

Option to define Bounding Box Stock for Mill part/assemblies using Pre-Defined Blocks

Implementation:

In previous versions of *CAMWORKS*, the option to define the stock for a mill part model/ assembly from a Bounding Box is available. Users could customize this stock by assigning the dimensions and offsets for the stock. From *CAMWORKS 2021* version onwards, an additional option for defining the bounding box stock with pre-machined blocks will be available. For such pre-machined blocks, the dimensions of the blocks are pre-defined. User can select one of the pre-machined bounding boxes from the available list of blocks. The stock can be easily aligned with the part using different options available within the *Stock Manager* dialog box.

These pre-defined stock sizes can be sorted based on the stock material group. Any changes to the pre-machined block can be saved into TechDB. Also, user can define the pre-machined blocks and assign them to a unique stock material group in TechDB.

~ 30	Ack manayer		-
Material : 1018		*	
Stock Type		*	
B	õ 🍞 🍪	6	
Co-ordinate Sy	stem	*	
Origin		\sim	
Stock Size		*	
Default	~		
Stock ID :	1018		
Length (X) :	5.5849in		
Width (Y) :	5.5849in	-	
Thickness (Z) :	0.375in	÷	
5.74	to Tach DR		
Save			
Stock Locate	c .	~	
X Reference :	Center		
Offset :	0in	-	
Y Reference :	Center	\sim	
Offset :	0in	▲ ▼	
Z Reference :	Center	\sim	
Offset :	0in	•	
Stock size		*	
X : 5.5849in	Y: 5.5849in Z:0.37	5in	
Description :	Default		
Number of Stoc	ks	*	
	Vorks NC Manager		



New - Option to Define Peck Amount for Point to point Operations in Denomination of Tool Diameter/Flute Length

Purpose:

Option to Define Peck Amount <u>for</u> Point to point Operations in the Denomination of Tool diameter or Flute Length

Implementation:

In previous versions of **CAMWORKS**, for Pecking cycles of Point to Point operations, the user could define the peck amounts (First peck; Sub peck amount, Minimum Peck Amount) in absolute values only. From *CAMWORKS 2021* version onwards, the option to define the peck amount with respect to the tool diameter or its flute length.

A button will be provided adjacent to each Peck amount value field in the Operation specific tab of Point to Point operations. Use this button to determine the method for assigning the Peck Amount Value. Following are the options available:

- [X] (Input Value): When this option is selected, the Peck amount will be defined using absolute values.
- **(% of Tool Diameter)**: When this option is selected, the Peck amount will be defined as a percentage of the assigned tool's diameter.
- **Q** (% of Flute Length): When this option is selected, the Peck amount will be defined as a percentage of the assigned tool's Flute Length.





Improved - Enhancements to Three Point Bore/Boss Methods for Probing Cycles

Purpose:

Option to modify parameters associated with Three Point Bore and Three Point Boss Probing Cycles

Implementation:

In previous versions of *CAMWORKS*, for the *Three Point Bore* and *Three Point Boss* methods of Probing cycles, the three points probed on the feature were always equidistant (120 degrees) from each other. No option was available to modify the angular distance.

From *CAMWORKS 2021* version onwards, this option has been introduced in the form of parameters within the **3** *Point Bore/Boss Parameters* group box under the **Probe** tab for a Probing operation. Parameters within this group box will be enabled only when the **Probe Cycle** method is set to **Three** *Point Bore, Three Point Boss* or *Automatic*.

Use the parameters in this group box to specify the angular distance between probe touch points.





Improved - Additional Parameters for Probing Cycles

Purpose:

Provide user interface for Probing Cycles so that users can specify additional probing parameters that are applicable only for certain Probing Cycles or specific use cases

Implementation:

In previous versions of **CAMWORKS**, all the Probing specific parameters were available under the **Probing** tab of Operation Parameters dialog box for Probing operations.

Under specific circumstances or for specific Probing cycles, it becomes necessary to specify additional parameters such as the tolerance of the feature being probed, the angular tolerance when measuring the angle between two faces, etc. Such parameters are applicable only in certain circumstances.

From *CAMWORKS 2021* version onwards, the user interface to assign values to all such additional Probe Parameters has been made available in the *Additional Probe Cycle Parameters* dialog box. This dialog box will be displayed when you click on the *More Parameters* button within *Probe Parameters* group box under the *Probe* tab.



'More Parameters' Button within 'Probe Parameters' group box under under Probe Tab

For all these parameters listed in the *Additional Probe Cycle Parameters* dialog box, you can choose whether the assigned values are to be post-processed or not. To output these values in the post-processed code, place a check in the checkbox to the left of the parameter's name. If unchecked, the value will not be post-processed.

Additional Probe Cycle Parameters		×
Additional Parameters	Description	
Angular Tolerance (Bb) :	1deg Null band. This is a tolerance zone in which	
Experience Value (Ee) :	0 no tool offset adjustment occurs.	t
% Feedback (Ff) :	0	
Feature Tolerance (Hh) :	0.01mm	
Position Tolerance (Mm) :	0.01mm	
Tool Offset (Tt) :	0	
Upper Tolerance (Uu) :	1mm	
Null Band (Vv) :	0mm	
Print (Ww) :	0	
<u>O</u> K <u>C</u> ancel	<u>H</u> elp	
Additonal Probe	Cycle Parameters Dialog Box	

New - Option to Specify whether Cutter Tool will roll over Surface Edges or be Clipped to Tool Contact Point

Purpose:

Checkbox option labelled *Waterfall Ends* to specify whether the toolpath will roll over surfaces edges or be clipped to the tool contact point

Implementation:

The *Waterfall Ends* checkbox option enables you to specify whether the toolpath will roll over surfaces edges or be clipped to the tool contact point. It will be enabled for only for *Constant Stepover* and *Pattern Project* operations when the *3 Axis toolpath generation method* is set to *Advanced Method* in *Update* tab of *CAMWorks Options* dialog box.

- When checked, the tool will roll over the edge of the surface a maximum distance of one tool radius.
- When unchecked, the toolpath will be clipped to the tool tangent contact point on the surface.

Note:

- For *Pattern Project* operations, this option will be enabled when the pattern in *Pattern* tab is set to any pattern except *Flowline*.
- For **Constant Stepover** operations, this option will be enabled only if the **By Slope** checkbox option in **Limits** group box under **Constant Stepover** tab is unchecked.







ool	F/S	Constant Stepover	Vostir NC	ig Links	Entry/Ret	Statistic: ract	s Advan	ced
Surface	finish							
cando	XY Allo	wance: Omm	÷					
	Z Allo	wance: 0mm	I					
	Avoid allo	wance : 0.1mm						
	Mach. de	viation : 0.01mm						
	Facet de	viation : 0.01mm			1			
Use	global facet d	eviation	1					
Directio								
	erse priority						1	
	croc priority			ľ				· .
Pass op	otions		\sim					
	Number	forte : 10	*					
	Contractor		 ▼ ▲ 	First	cut from : T	fon of St	tock	\sim
	Constant c	utamt: 2.5mm	-		0"			
Rev	erse order				Uniset : U		•	
Axial of	fsetting	<i>c</i> , 1		Li	ast cut at : E	Bottom o	f Stock	\sim
	Number				Offset : 0	mm	* *	
	Axia	loffset : Jamm	-	By slope				
					Min angle : 0)deg		- -
				M	Max angle : 9	0deg		Ŧ
				Contact areas	s only			
				✓ Waterfall end	s			
		_						



New - Option to Recognize Counterbore Hole Features as Simple Hole Features

Purpose:

Option to recognize steps of the Counterbore holes as individual hole features

Implementation:

Within the *Hole Recognition Options* group box under the *Mill Features* tab of the *CAMWorks Options* dialog box, a new checkbox option named *Recognize Counterbore Holes* has been introduced. This checkbox option will be checked by default.

era Mill Features Display Simulation Up	date File Locations
Facet deviati	on: 0.0127mm
Spline deviati	on: 0.0254mm
xtract machinable features	Method · MfaView
Feature types	Remove on rebuild
✓ Holes	Holes
✓ Non holes	Non holes
Boss	Boss
Face	
Part perimeter	Part perimeter options
✓ Tapered & filleted	◯ Open pocket type
Multi surface pockets	Boss type
Curve features for chamfering	Local features
Hole recognition options	Smart pick
Max diameter : 25mm	Adjacent faces
Min included angle : 210.00deg	Curve feature options
Condense split holes	Max face angle : 15.00deg
Recognize counterbore holes	Planar edges only
Extend holes to the stock Apply.	

- If this option is checked, all hole features with counterbore shape will be recognized as counterbore holes.
- If this checkbox option is unchecked, then during *Automatic Feature Recognition* of a hole feature with counterbore shape, each step will be recognized as individual simple holes. (Refer the illustration below.



Note:

If the diameter of the any of the recognized simple holes exceeds the *Max diameter* defined in the *Hole Recognition Options* group box, then it will be recognized as a circular pocket feature.



New - Recognize Pockets and Slots having Non-Uniform Top or Bottom Fillets or Chamfers

Purpose:

Improvise *Automatic Feature Recognition* such that 2.5 Axis Mill Features having non-uniform top and/or bottom fillets, chamfers or drafts are recognized

Implementation:

In previous versions of *CAMWORKS*, when *Automatic Feature Recognition* was executed via the *Extract Machinable Features* command, only 2.5 Axis Mill features having uniform geometry were recognized.

Note:

2.5 Axis Mill Tapered features will be recognized only if the Tapered and Filleted option within Features Types group box under Mill Features tab of Options dialog box is checked.

From **CAMWorks 2021 SP0** version onwards, the **Automatic Feature Recognition** functionality will be enhanced such that additional 2.5 Axis Mill pocket and slot features will be recognized in the areas of the part model having non-uniform fillets, chamfers, or drafts. Such pocket and slot features will be recognized subject to the following conditions:

- i. The bottom of the pocket or slot must be flat and normal to the machining direction.
- ii. Any non-uniform fillets (i.e. a fillet with variable radius and height) on the feature to be recognized will be ignored.



6

- iii. Any non-uniform chamfers (i.e. a chamfer with variable angle and size) on the feature to be recognized will be ignored.
- iv. If a feature on the part model has a surface with non-uniform draft, then the feature will be recognized without considering the surface with draft.





Technology Database

New - Assigning Default Feature Options for Part Perimeter Features in TechDB

Purpose:

Provision to assign default parametric values for Part Perimeter Features in TechDB

Implementation:

There are two feature types for the **Part Perimeter** feature viz. **Perimeter feature (Open pocket)** and **Perimeter feature (Boss)**.

In previous versions of CAMWorks, for both these Part Perimeter feature types, there was no provision to assign different default end conditions for the different Mill Setups of the part model/ assembly.

From the **CAMWorks 2021 SPO** version onwards, a new user interface named **Default Feature Options** user interface has been introduced in TechDB under the **Strategies** sub-menu of **Mill** menu. Use this interface to define, edit, view and/or delete the End Conditions for the Part Perimeter feature types. Multiple End Conditions can be defined but only one of them can be assigned as the default. The presence of a default end condition for Part Perimeter feature in TechDB further automates the insertion of Part Perimeter feature within the CAMWorks user interface.

Within CAMWorks, when inserting a new Part Perimeter feature via *Automatic Feature Recognition* (AFR) or *Interactive Feature Recognition (IFR)*, the default End Conditions for the part perimeter feature being inserted will be determined by the default values assigned within this user interface of TechDB.

	Mill > Default Feature Options	Metric Inches 3
Mill	Feature Perimeter feature (Open pocket)	
Turn	Id Perimeter feature (Boss) ≡ Sa 1 Perimeter fe Default ✓	ve Copy Delete eneral (ID: 1)
🔒 Mill-Turn		Default : 🗹 Description : Default
EDM		End condition options FCS -ve Z Axis : Bottom of the stock
Mill Tooling		Offset : 0 in FCS +ve Z Axis : Top of the part
🛃 Turn Tooling		Offset : 0.39 in Other direction : Bottom of the part v
Feed / Speed		Offset : 0 in
'Def	ault Feature Options' User Interface in Mill Menu	of Technology Database



New - Option to Re-sequence Operations in the Features & Operations User Interface

Purpose:

To provide an option to re-sequence operations listed in the Operations table for *Feature and Operations* user interface and other user interfaces where operation sequences have been defined

Implementation:

Within the *Technology Database* application, the *Feature and Operations* user interfaces for Mill, Turn and EDM and other user interfaces (such as Multi-stepped Holes, Thread Mill, Tap Cutting, Tap Rolling) are used to define/edit operation sequences that will be used to machine the different machinable features. You can create and store machining sequences of repetitive operations.

For a selected entry within these user interfaces, the corresponding operation sequence will be displayed in the *Operations* table.

If no operation sequence has been defined for a Feature Condition, then this table will be empty. The commands available for the *Operations* table are to insert operations and create an operation sequence.

In previous versions of CAMWorks, there were no options available within this user interface to resequence the operations listed in the **Operations** table. Consequently, users could neither insert any operation between two listed operations nor change the sequence.

From **CAMWorks 2021** version onwards, the **Move Up** and **Move Down** command buttons have been made available in the **Feature and Operations** user interfaces. If two or more operations are listed in the **Operations** table, you can use the **Move Up** and/or **Move Down** command buttons to rearrange the operation sequence.

In Technology Database, the *Move Up* and *Move Down* command buttons will be available in the following user interfaces for the resequencing operations.

- Mill>>Strategies>>Features & Operations
- Mill>>Strategies>>Thread Mill
- Mill>>Strategies>>Multi-stepped Holes
- Turn>>Strategies>>Features & Operations
- Turn>> Strategies>>Thread Condition Operations
- EDM>> Strategies>>Features & Operations
- Mill Tooling>>Threading Strategies>>Tap Cutting Strategies
- Mill Tooling>> Threading Strategies>>Tap Cutting Strategies
- Turn Tooling>>Threading Strategies>>Tap Cutting Strategies
- Turn Tooling>> Threading Strategies>>Tap Cutting Strategies

d	Operation	Tool Selection	Depth Criteria	V
567	Center Drill	(Diameter + 0.5) & (Diameter + 5)	(Diameter * 0.9)	M
577	Drilling	(Diameter - 0.4) & (Diameter - 0.3)	(Feature Depth + 0)	M
578	Contour Mill	(Diameter - 5) & (Diameter - 2)	(Feature Depth + 0)	M
579	Boring	(Diameter + 0) & (Diameter + 0)	(Feature Depth + 0)	M

Move Up and Move Down Buttons in Feature & Operations User Interfaces



New - Functionality to Import Tools from Harvey & Helical Tools Catalog into TechDB

Purpose:

Option to import Mill Tools from Harvey and Helical Tools Catalog into TechDB

Implementation:

From CAMWorks 2021 version onwards, the tool catalogs of Harvey and Helical Tools (both owned by Harvey Performance Company) will be shipped along with CAMWorks. After CAMWorks is installed, these tool catalogs will be available in JSON (JavaScript Object Notation) format at the following location:

C:\CAMWorksData\CAMWorks202Xx64\Tooling\Harvey Helical

The JSON format is an open standard file format as well as a data interchange format. It therefore allows transmission of data objects consisting of attribute-value pairs, array data types and serializable values. The tool catalogs of Harvey and Helical Tools (in JSON format) primarily contain End Mill and Taper End Mill tools. These tools can be imported into TechDB using the corresponding user interfaces for these tools.

These following command options to import Harvey and Helical tools will be available under the

Options for Importing/Exporting Info Displayed in Tables menu in the user interfaces for End Mill Tools and Taper End Mill Tools in the TechDB.

- Import Harvey Tool 🚏
- Import Helical Tool 🚏 •

Executing either of these commands displays the *Import Tools* window. Use the parameters and options given within this window to import the desired Harvev/Helical tools into TechDB.

🕝 CAMWorks 2021 Techno	ology Da	atabase										-	đ	×
	С	Mill Tool	ling > Cutters									Metri	c Inches	0
Aill Mill	Flat	End Mill					¥	Ø	T	Save Copy Delete	•			
	Id	Active	Tool I D	Sub-Type	Shank Type	Shoulder	0		9 Exp	portCSV (all fields)	2			
📴 📶 Turn	19	1	1/64 EM CRB 2	Rough & Finish	Straight	0.015625	1.		9 Exp	portCSV (Current fields)		4.00		1
	20	~	1/32 EM CRB 4	Rough & Finish	Straight	0.03125	1.		. Inst		1/64 EM CRB 2F	L 1/32		
Mill Turn	21	~	1/32 EM CRB 2	Rough & Finish	Straight	0.03125	1.	111	×	;	Rough & Finish	*		
	22	~	3/64 EM CRB 4	Rough & Finish	Straight	0.046875	1.		🦻 Imp	port - Harvey Tool	Straight	•	L2 🗸	/
•	23	~	3/64 EM CRB 2	Rough & Finish	Straight	0.046875	1.		🦻 Imp	port - Helical Tool	0.015625 in		<u>↓↓</u> <i>µ</i>	4
EDM	24	~	1/16 EM CRB 4	Rough & Finish	Straight	0.0625	1.			Shank dia. (DZ)	0.125 in		00	· •-
	25	~	1/16 EM CRB 2	Rough & Finish	Straight	0.0625	1.	5		Shoulder Dia (D4) :	0.015625 in			
Mill Tooling	26	~	5/64 EM CRB 4	Rough & Finish	Straight	0.078125	1.	5		Overall length (L1) :	1.5 in			
	27	1	5/64 EM CRB 2	Rough & Finish	Straight	0.078125	1.	5		Flute length (L2) :	0.03125 in			
л	28	1	3/32 EM CRB 4	Rough & Finish	Straight	0.09375	1.	5		Shank Length (L6) :	1 in			
Turn Tooling	29	1	3/32 EM CRB 2	Rough & Finish	Straight	0.09375	1.	5		Protrusion (L3)	0.00275			
	30	1	7/64 EM CRB 4	Rough & Finish	Straight	0.109375	1.	5		Shoulder length (L4) :	0.09375 In			
鲸 Feed / Speed	31	1	7/64 EM CRB 2	Rough & Finish	Straight	0.109375	1.	5		Shoulder lengul (L4).	1 in			
+-4	32	1	1/8 EM CRB 4F	Rough & Finish	Straight	0.125	1.	5		Hand of cut :	Righthand	*		
	33	1	1/8 EM CRB 2F	Rough & Finish	Straight	0.125	1.	5		No. of flutes :	2			
Settings	34	1	9/64 EM CRB 4	Rough & Finish	Straight	0.140625	2			Center Cutting :				
-	35	~	5/32 EM CRB 4	Rough & Finish	Straight	0.15625	2			Tool material :	Carbide	•		
(j) About	36	~	5/32 EM CRB 2	Rough & Finish	Straight	0.15625	2			Comment :	1/64 EM CRB 2F	L		
~	37	1	11/64 EM CRB 4	Rough & Finish	Straight	0 171875	2				1/32 LOC	/		

Option to Import Harvey & Helical Tools in End Mill & Taper End Mill User Interface of TechDB

Tools Types from Harvey & Helical Tools Catalogs can be Imported into TechDB

Tools from the tool catalogs of Harvey and Helical Tools (in JSON format) can be imported into TechDB for the following Mill Tools using corresponding user interfaces:

- Flat End Mill Tools
 - Ball Nose Mill Tools
- Taper Flat End Mill Tools
- Hog Nose Mill Tools
- Taper Ball Nose Mill Tools



New - Option to Modify Post Processor Path in Technology Database

Purpose:

To provide an option within the TechDB app to the user to set the path of a folder to select the post processors.

Implementation:

How the Assignment of Post Processor to a Machine is done in Previous CAMWORKS Versions

Within the *CAMWORKS* application, users can assign the Post Processor to be associated with a Mill or Turn machine in the *Post Processor* tab of the *Machine* dialog box. Using the Browse button within this tab, users can set a folder path to a specific folder. All post processors within the folder and sub folders thus specified will become available for selection in the *Available* list box within the *Post Processor* tab. To select the desired post processor, the user can highlight it in the *Available* list box and click the *Select* button.

Within the TechDB linked to the *CAMWORKS* application, the option to assign/reassign the post processor associated with the active machine is available in the General tab of that active Mill Machine/ Turn Machine. This option is provided in the form of the Post Processor dropdown list. It will list the same post processors present within the folder specified in the *Post Processor* tab of *Machine* dialog box in *CAMWORKS* application. (The listed post processors also include those within the sub-folders of the specified folder.)

The current limitation in previous *CAMWORKS* versions is that users cannot specify any alternate folder in TechDB from where post processors can be made available for selection when defining or editing Mill or Turn machines. The selection can be made strictly only from the folder defined in *CAMWORKS* application.

Mill > Machines		Metric Inches
Mill - Inch (Default)	Save Copy Delete	~
Mill 4 axis - Inch	Default Machine :	Ø
Mill 5 axis - Inch	Machine name :	Mill - Inch
	Machine ID :	Milling Machine Inch
	Description :	Sample Milling Machine
	Post Processor :	M3Axis-Tutorial.CTL v
	Machine Duty :	M3Axis-Tutorial.CTL
	Default Feature Strategies :	M4AXIS-TUTORIAL.CTL M5AXIS-TUTORIAL CTI
	V Subroutines	MT2AXIS-SIEMENS-TUTORIAL.CTL
	Output subroutines for Mill Operations	MT4AXIS-TUTORIAL.CTL
	Output subroutines for feature patterns and part instances	T2AXIS-TUTORIAL.CTL
		T4AXIS-TUTORIAL.CTL
	(Part Mode: Output subroutines)	MILLACRAMATIC-2100.CTL
	Assembly mode.output subjournes	MILLALLENBRADLET-6400.CTL
	Tool	MILLANILAM_6000M.CTL
	Feature	MILL/BOSTOMATIC.CTL
	Part	MILL\BROTHER-C720.CTL

Post Processor Dropdown List in Mill Machines UI of the Technology Database App

Introduction of Option to Set Post Processor Path in SOLIDWORKS 2021 version

From SOLIDWORKS 2021 version onwards, the option to set the folder path for post processor selection has been introduced in the **Settings tab** of the Technology Database UI. This is provided in the form of the Post Processor path field and associated Browse button. Under default settings, this path will be identical to the folder path specific in the *Post Processor* tab of *Machine* dialog box in *CAMWORKS* application. To change the path, click on the **Browse** button and use the **Browse for Folder** dialog box that will be displayed to select the desired folder location Once the new folder location is selected, the **Post processor path** field will be updated to display the new folder path. In



the **General** tab of the **Mill Machine** and **Turn Machine** interface within the TechDB, the **Post Processor** dropdown list will now display post processors present within the currently selected folder and its sub-folders.

=	C Settings	Metric Inches ?
Mill	General Application Default : Mill	v v
📴 🗐 Turn	Post Processor Path : C:\CAMWorksData\CAMWorks2020x64\posts	
Mill-Turn	Automatic : 🕑	~
EDM	Language : English / English	T
Mill Tooling	Customization Settings This functionality allows you to save and restore customization settings for TechDBApp grid column Save Settings Restore Settings	n visibility and order location.
Jurn Tooling	Link Database	Import Database
Feed / Speed	It is recommended to ensure SOLIDWORKS is not running before proceeding further SQLite Ms-Access SQL Server Please choose the location of the source database. Browse C:\CAMWorksData\CAMWorks2021x64\TechDB\TechDB.cwdb	:
Post Proce	essor Path and Browse button in Settings UI of the Technology	y Database App



Mill-Turn

New - Support for Probe Tools and Probing Operations extended to Mill-Turn Mode

Purpose:

To extend the functionality of Probing Operations using Probe Tools available in Mill mode to Mill-Turn mode

Implementation:

Probing is an established best practice for maximizing the efficiency, quality, capability, and accuracy of machine tools.

This functionality was introduced in the CAMWorks 2020 SP0 version for Mill mode.

From *CAMWorks 2021 SP0* version onwards, this functionality has been extended to the Mill-Turn mode too. Probing operations can be added under Mill Part Setups in the Mill-Turn mode. The user interface related to the Probing operations will be identical to those available in the Mill mode.

	New Operation 🥥	Í
New Setup	*	
1		
✓ Associate		
Pick from the Exist	ing 🕅	
Mill Part Setup1		



Multiaxis Mill

Improved - Additional Parameters for Multiaxis Milling Operations

Purpose:

Provide additional parameters for Multiaxis Milling operations to ensure additional flexibility when defining those operations

Implementation:

From CAMWorks 2021 version onwards, the following additional parameters will be available in the various tabs of the Multiaxis Mill operations:

Gouge Checking Tab for Multiaxis Milling

Within this tab, new checkbox option for minimizing titling will be available. This checkbox will be enabled when the following conditions are fulfilled:

- Gouge Checking Strategy is set to Tilt tool •
- Angle Definition is set to **Automatic** •
- The option selected in the **Preferences** group box is set to **Equal tilting and rotary**

Sta	tistics	Axis Control	Finish	Deugleing	Re	est
Tool	F/S	Pattern Entry/F	Retract Links	Gouge Checking	Advanced	Posting
Option	s					
🗹 Ch	ieck gouges b	etween positions	Holder clearan	ce : 1mm 🔺	÷	
Ex	tend tool to inf	inity	Shank clearan	ce : 1mm	+	+
Ch	eck link motio	ns for collision	Non-outting clears			
Ch	eck tip radius				L 11.	
Stock	collision chec	sking <mark>Shaft</mark>	Arbor	der		
Group	0 1 Group 2	Group 3 Group 4				
- 6-	nuga abaakina		- Choole pariod	·		
	Apply aquae	checking to		urfaces		
-	Holder		Other surf	aces		
	Shank		Allowance	0mm		
	Non-cuttin	g portion 🗧	Allowance.			
	✓ Flute	2				
G	ouge check op	otions				
	Strategy	: Tilt Tool	~	_		
Ar	nale definition	Automatic	~ >			
	- Datany avia	7 A.:.				
		. Z AXIS	~			
	rieleiences	Equal tilting and results	otary			
		Prefer tilting				
		 Prefer tilting Prefer rotary 	-			





Multiple Cuts Tab for Multiaxis Swarf Milling

Within this tab, the new Sorting Method of Spiral will be available. When this option is selected, CAMWorks will apply a spiral method to the stepdown and stepover passes. This method is only available for Stepdown when *Pattern* is set to *Morph* and *Direction* option of *Along Tool Surface* is not used.

	F (0	D	E 1 (D 1 - 1	1.1	0 0 I			D. 11
00l	F/S tatistics	Pattern	Axis Control	Links	Multiple Cuts	ng Adv	/anced Co	Posting
-								
Stepdo	wn Pattern —							
	Depth steps :	By Number	of Slices \checkmark					
Nur	nber of slices :	3						
	Pattern :	Morph	~					
	Direction :	Along Tool	Axis 🗸	6				
					4		1	
Tool Gu	lidance							
	Damping :	None	~					
	Damping dist :	0in	×		0			
	Tool shift :	Constant for	each Slice 🗸 🗸					
	Start :	0in	* *					
	End :	0in						
Stepov	er Pattern							
Nu	imber of cuts :	1						
Sorting								
Johang	Matheast	0 : 1						
	Method :	Spiral	~					

'Spiral' Method for Sorting in Multiple Cuts Tab for Multiaxis Swarf Milling Operation

Entry/Retract Tab for Multiaxis Swarf Milling

Within this tab, the **Tool Axis Orientation** dropdown list parameter will be enabled in the Leadin and Leadout pages when Type is set to Automatic Arc. Additionally, a new option **Automatic** will be available in the **Tool Axis Orientation** dropdown list.

When the *Automatic* option is selected in the *Tool Axis Orientation* dropdown list, CAMWorks will automatically determine the Tool orientation along the leadin/leadout arc. This option is only available when the leadin/leadout Type is set to *Automatic Arc*.



Stati	stics	Axis	`ontrol		Finish	R	oughing		Rest
Tool	F/S	Pattern	Entry.	/Retract	Links	Gouge Che	cking	Advanced	Posting
— Leadin I	nove								
Fr	m · Clearar	nce	~						
Moth	od : Use Le								
		adin	~						
Star	from home	position							
Leadour	tmove			Home p	osition	al			
	To: Clearar	nce	\sim	X: 0m	m 🔹				
Meth	od : Use Le	adout	\sim	Y: 0m	m 🖡		F		~
Retu	um to home j	position		Z: 0m	m 🔹			1	
Clearan	се								
Ту	pe : Plane I	n Z		~					
াল্ল	7.00mm	·			Leadin [.eadout			
42	2 : John	•							
						Type :	Automati	c Arc	~<
					Tool axi	s orientation :	Automati	c	~
					Param	eters			`



Turn/ Mill-Turn

New - Option to retain Arc Moves for B Axis Continuous Turn Toolpaths

Purpose:

To provide the option to users to retain arc moves for B Axis Continuous Turning Toolpaths

Implementation:

In previous versions of CAMWorks, when Continuous B Axis Turning toolpaths were generated, any arc moves present were converted to line moves.

From **CAMWorks 2021 SP0** version onwards, the option to retain or not retain arc moves will be provided in the **Advanced** tab for **Turn Finish** operations in the form of the **Convert arcs to lines** checkbox option.

The group box named **Spline Output** present under **Advanced** tab for Turn operations has been renamed to **Lines and Arcs**. The newly introduced **Convert arcs to lines** checkbox option will be available within this group box.

How the 'Convert arcs to lines' Checkbox Option will work

This checkbox option will be enabled only when the **Continuous B Axis** checkbox within **Continuous B Axis Turning** group box under **Turn Finish** tab for a **Turn Finish** operation is checked. This checkbox will be checked by default. It will be disabled for Turn operations other than *Turn Finish*.

- When unchecked, arc moves generated for the *Continuous B Axis Turn* toolpath will be retained and not converted to linear moves.
- When checked, arc moves generated for the *Continuous B Axis Turn* toolpath will be converted to linear moves.

peration Parameters	1	- 0	×
Fool F/S Turn Finish NC Lead In/Out	Feature Option Advanced	tatistics Posting	
Z limits Z end To: WIP Z end : Omm Use setup definition Z start From : WIP Z start : Omm Use setup definition			z Lines and arcs
Unes and arcs Convert arcs to lines Arc fit spline facets Spline deviation : 0.03mm	Chuck/Fixture avoidance Enable Clearance : 2mm Use setup definition	1 D	Convert arcs to lines Convert arcs to lines Arc fit spline facets Spline deviation : 0.03mm

Lines and arcs' group box with 'Convert arcs to lines' checkbox option under Advanced tab for Turn Finish Operations



New - Assembly Mode Support for Turn and Mill-Turn

Purpose:

To extend the Assembly support function of Mill mode to Turn and Mill-Turn modes of CAMWorks

Implementation:

From *CAMWorks 2020 SP3* version onwards, Assembly mode functionality will be available for Turn and Mill-Turn mode of CAMWorks. In this Turn/Mill-Turn Assembly mode, assemblies containing a max of two-part models will be supported.



Sample Image of an Assembly comprising Turn and Mill-Turn parts

Defining Coordinate System for Machining the Turn/Mill-Turn Assembly

Use the Machine tab of the Machine dialog box to select the Machine that will be used to machine the part models constituting the Turn/Mill-Turn assembly. Users will have to define the Main Spindle and (optionally) Sub Spindle Coordinate system for this machine. These Coordinate Systems will be used to define the Turn axes of the machine. The Turn axes defined for both the spindles must be co-axial. Use the *Edit Definition* button in the *Setup* tab of the *Machine* dialog box to invoke the *Main Spindle/Sub Spindle Coordinate System* dialog box. Alternatively, you can double-click on the *Coordinate System* node under expanded *Machine* node I^A in CAMWorks Feature tree and Operation tree.

For each part model, the origin of the Main Spindle and (optionally) Sub Spindle Coordinate System must be separately defined. The origin can be defined using an entity, part vertex or stock vertex. You can double-click on the *Coordinate System* node in under expanded Turn part nodes listed under *Part Manager* node in CAMWorks Feature tree and Operation tree to invoke the *Main Spindle/Sub Spindle Coordinate System* dialog box.

Defining Stock for the Assembly

Stocks must be defined for each part that constitutes the assembly. The stock for the individual parts will be listed as sub-nodes under Stock Manager node in the CAMWorks Feature tree/ Operation tree. Double-clicking on the any of these **Stock Manager** nodes displays the **Stock Manager** dialog box.

Use this dialog box to define/edit the stock. Use the *Number of Stocks* group box within the *Stock Manager* dialog box to select the part model for which stock is to be defined.



Coordinate System Nodes under Machine node and Turn Part nodes in CAMWorks Operation tree





Manage Parts

The Manage Parts dialog box is displayed when you double-click on the **Part Manager** node **b** in the Feature tree/Operation tree. This dialog box allows you to:

- i. Select the Turn/Mill-Turn parts of the active assembly to be machined
- ii. Assign the Spindle Designation for each selected part
- iii. Assign the Origin of the Coordinate System associated with the Main Spindle/ Sub Spindle for each selected part
- iv. Edit the Turn Feature Section Plane

Manage Parts	
Selected Parts :	Spindle designation Main Spindle Sub Spindle Both
 Origin Main spindle co-ordinate system Sub spindle co-ordinate system 	Turn feature section plane Modify : Type : XZ Plane Entity : Angle : 0.000deg
ОК	<u>H</u> elp

Manage Parts Dialog Box

Defining Programmable Offsets for Main Spindle & Sub Spindle of Active Machine

Programmable coordinate offsets will have to be specified for the Main Spindle and Sub Spindle of each Turn/Mill-Turn part in the assembly.

Click on the *Edit* button within the *Work Offset* group box under *Setup* tab of *Machine* dialog box to view the *Spindle Work Coordinate* dialog box. Use the parameters within this UI to specify/edit the programmable offsets Main Spindle and optionally Sub Spindle of the active machine.

Work Coordinate will be the default method for defining the offsets. For each part in the Turn/Mill-Turn Assembly, the default Main Spindle value will be set to 54 and the default Sub Spindle value will be set to 55. These default values can be reassigned to suit your requirements.

Spindle work coordinate offset		
	Main	Sub
○ <u>N</u> one		
○ <u>F</u> ixture	1	1
<u>W</u> ork Coordinate	54	55 🜲
⊖ Work & <u>S</u> ub Coordinate	1	1
	I	I



New - Option to Define the Spindle Direction as Clockwise for Turn and Mill-Turn Machines

Purpose:

To provide a mechanism whereby users can set the Spindle Direction (for Turn and Mill-Turn machines) as Clockwise relative to the Machine Operator

Implementation:

From **CAMWorks 2021** version onwards, a new functionality that enables users to set the Spindle Direction as Clockwise for the Main and Sub Spindles of the active Turn/Mill-Turn machine has been introduced. The reference to set the spindle direction as clockwise will be relative to the Machine operator.

Functionality to Define the Spindle Direction as Clockwise for Turn & Mill-Turn Machines

This functionality is provided in the form of the *Clockwise Spindle Direction* dropdown list for Main Spindle and Sub Spindle within *Spindles* group box of the *Setup* tab in *Machine* dialog box.

- If the clockwise rotation of the Main spindle and/or Sub spindle is the direction that rotates away from the operator, then select the *Away from Operator* option in the corresponding dropdown list.
- If the clockwise rotation of the Main spindle and/or Sub spindle is the direction that rotates towards the operator, then select the *Towards Operator* option in the corresponding dropdown list.

Machine Tool Crib Post	Processor Posting	Setup	Chuck/Fixture		
Spindles					
₽	Main spindle		Sub spindle	•	
Coordinate system :	🙏 User Define	ed	4.	User Defi	ìned
Work offset :	[None]			[None]	
Spindle speed : 🗆 l	limit		Limit		
450	0.0000rpm		4500.0000)rpm 🚔	
C.	Clockwise :		Clockwise :		<i>.</i>
Spindle direction :	Towards Operato	or ~	Away from	Operator	~ 41
	Ass		inting operation		1
	,	sociate ex	isung operations	s	
		sociate exi	sung operation:	· · · ·	
Tum feature section p		um feature	e display plane	ə	
Turn feature section p Type : XZ Plan	lane Tu	um feature Type	e display plane : XZ Plane	•	
Tum feature section p Type : XZ Plan Entity :	lane Tu	um feature Type Entity	e display plane : XZ Plane :	×	
Tum feature section p Type : XZ Plan Entity : Angle : Odeg		um feature Type Entity Angle	e display plane : XZ Plane : .	· · · · · · · · · · · · · · · · · · ·	
Tum feature section p Type : XZ Plan Entity : Angle : Odeg	lane Tu	um feature Type Entity Angle	e display plane : XZ Plane : : : : : Odeg		
Tum feature section p Type : XZ Plan Entity : Angle : Odeg	lane Tu e V I G-code coordinates	um feature Type Entity Angle	a display plane : XZ Plane : Odeg		
Tum feature section p Type : XZ Plan Entity : Angle : Odeg Options Display toolpath at	lane Tu e V i G-code coordinates omp on first move	um feature Type Entity Angle	e display plane : XZ Plane : Odeg	· · · · · · · · · · · · · · · · · · ·	
Tum feature section p Type : XZ Plan Entity : Angle : Odeg Options Options Display toolpath at	lane Tu e V i G-code coordinates omp on first move	um feature Type Entity Angle	e display plane : XZ Plane : Odeg		





Resetting the Spindle Directions of Turn Operations for Legacy Turn/Mill-Turn Parts

If you wish to reset the user-assigned spindle directions of Turn operations for legacy parts, then click on the **Associate Existing Operations** button in the **Setup** Tab of **Machine** dialog box. Clicking on this button displays the **Dialog box for Turn Operations with Incorrect Spindle Directions**. This dialog box lists all Turn operations for which the spindle directions are with reference to settings made on the machine node.

- Click on the **Yes** button within this dialog box to rectify the spindle directions of the listed Turn operations. This action will auto-assign the Spindle direction in the *F/S tab* for the Turn operations based on parameters that control the spindle direction. The *Override Spindle Direction* checkbox in the *F/S* tab of operation parameters will be unchecked.
- If you click **No** within this dialog box, then the user-assigned spindle directions for all the Turn operations listed within the dialog box will be retained. The **Override Spindle Direction** checkbox option in the **F**/**S** tab of operation parameters will continue to remain in checked state for these Turn operations.

Do you want to correct the spindle direction for the following? Operation name Current Spindle Direction RearMainRightFaceDown Le CW RearMainRightFaceLeft UpN CW RearMain LeftFaceLeft UpYe CW RearMain LeftFaceLeft UpYe CW	direction as per the tool and fea	ature combination.
Operation name Current Spindle Direction RearMainRightFaceDown Le CW RearMainRightFaceLeft UpN CW RearMain LeftFaceLeft UpYe CW RearMain LeftFaceLeft Down CW	o you want to correct the spin ollowing?	dle direction for the
RearMainRightFaceDown Le CW RearMainRightFaceLeft UpN CW RearMain LeftFaceLeft UpYe CW RearMain LeftFaceLeft UpYe CW	Operation name	Current Spindle Direction
RearMainRightFaceDown Le CW RearMainRightFaceDown Le CW RearMainRightFaceDown Le CW RearMainRightFaceLeft UpN CW RearMain LeftFaceLeft UpYe CW RearMain LeftFaceLeft UpYe CW	RearMainRightFaceDown Le	CW
RearMainRightFaceDown Le CW RearMainRightFaceDown Le CW RearMainRightFaceLeft UpN CW RearMain LeftFaceLeft UpYe CW RearMain LeftFaceLeft Down CW	RearMainRightFaceDown Le	CW
RearMainRightFaceDown Le CW RearMainRightFaceLeft UpN CW RearMain LeftFaceLeft UpYe CW RearMain LeftFaceLeft Down CW	RearMainRightFaceDown Le	CW
RearMainRightFaceLeft UpN CW RearMain LeftFaceLeft UpYe CW RearMain LeftFaceLeft Down CW	RearMainRightFaceDown Le	CW
RearMain LeftFaceLeft UpYe CW RearMain LeftFaceLeft Down CW	RearMainRightFaceLeft UpN	CW
RearMain LeftFaceLeft Down CW	RearMain LeftFaceLeft UpYe	CW
< >	RearMain LeftFaceLeft Down	CW >

Dialog Box for Turn Operations with Unsynced Spindle Directions listing the name of Turn Operations with Spindle Directions that are not in sync as per the Tool and Feature Combination



New - Mutual Exclusivity of Order of Operations listed in Sync Manager & Operation Tree

Purpose:

- i. Option to re-position operations in *CAMWorks Sync Manager* user interface independently without restrictions imposed by operation sequence in Operation tree
- ii. Provide command buttons to execute Step Through toolpath and Post Processing from the CAMWorks Sync Manager user interface

Implementation:

For part models and assemblies machined using dual-turret or multi-turret Turn/Mill-Turn machines, the *CAMWorks Sync Manager* functionality allows you to synchronize the toolpaths machined by the tools in the front and rear turrets. This is achieved by re-ordering the operations in the *Sync Manager* tab of the *CAMWorks Sync Manager* user interface. Optimal synchronization can drastically reduce the time-required to machine the parts/assemblies

In previous versions of CAMWorks, the flexibility is re-ordering operations machined using tools in front and rear turrets was limited within the *CAMWorks Sync Manager* user interface as the sequence of operations was linked to the sequence of operations and setups in the Operation tree. For optimal operation sequencing, users had to manually split and/or reorder setups. This often resulted in bloated Setups and a confusing sequence of operations.

From **CAMWorks 2021 SP0** version onwards, operations can be re-sequenced in the Sync Manager tab of the CAMWorks Sync Manager user interface independent of their order in the Operation tree. When the Syncing Strategy is set to User Defined, all the operations listed from the CAMWorks Operation tree will be listed in their respective turret columns within the Sync Manager tab. To reorder the sequence of machining for syncing operations, you can drag and drop the operations to the desired locations in the turret columns as follows:

- By dragging and dropping a selected operation to the desired location within its turret column
- By dragging and dropping it to the specific desired location of another turret column (After re-location, the toolpath of the selected operation will be regenerated.)

Note:

If the position of any operation in the turret column is changed, then all the toolpaths for the operations listed in the turret columns will be regenerated without any warning message being displayed. This behavior is observed because, by default, the **Update toolpaths when parameters change** option in the **Update** tab of **CAMWorks Options** dialog box is set to '**Always**'. To receive a warning message regarding the recalculations of toolpaths when operation parameters change, set the **Update toolpaths when parameters change** option to '**Prompt**'. This setting ensures that whenever the position of an operation in the turret is changed, CAMWorks will display a warning message regarding the need to regenerate toolpaths. Click **Yes** within the warning message to regenerate the toolpaths.

Executing Step Through Toolpath Command 🔤 from CAMWorks Sync Manager User Interface

The **Step Through Toolpath** command is now available in the *CAMWorks Sync Manager* user interface. If you execute this command, then the *Step Through Toolpath* dialog box will be displayed. Use this user interface to step through the toolpaths. The toolpath sequence for stepping through the toolpaths will be based on the order of operations listed in the *Sync Manager* tab. It will not be based on the operation sequence listed in the Operation tree.



The **Post Process** command is now available in the CAMWorks Sync Manager user interface. If you execute this command, then the NC program generated will be based on the order of operations listed in the Sync Manager tab. It will not be based on the operation sequence listed in the Operation tree under various Turn or Mill Setups.

CAMWorks Sync Manager	×
Sync Manager Time View	
Syncing strategy User defined By operation By turret	
Rear Turret 1	Front Turret 1
 Face Finish1 [T01] Center Drill1 [T02] Groove Finish1 [T04] Totol Groove Finish2 [T04] Groove Finish3 [T04] Groove Finish4 [T04] Groove Finish6 [T04] Groove Finish6 [T04] M Groove Finish6 [T04] M Rough Mill3 [T10] Post Operation2 Cut Off1 [T04] 	
	OK Cancel Help
Sync Manager Tab of CAMWorks	Sync Manager User Interface



Feed & Speed Library

Improved - Updated 64-bit Feed and Speed Library now Shipped with CAMWorks

Purpose:

To provide an enhanced version of the Feed and Speed Library with SQL-based database engine to users

Implementation:

In previous versions of CAMWorks, a 32-bit version of the *MES Feed and Speed Library* was shipped along with CAMWorks. This application had a MS Access based Database Engine.

From **CAMWorks 2021 SPO** version onwards, the 64-bit version of the **MES Feed and Speed Library** will be shipped in the CAMWorks Installer Package. This application has an SQL-based database engine. For users who wish to further customize the entries present within the *Feed and Speed* library, the user interface too has undergone a makeover that makes the navigation and interaction intuitive and smooth.

If you had customized feed and speed data in the 32-bit Feed and Speed Library, then you will need to import it into the 64-bit *Feed & Speed Library*. Given below are the steps for importing customized data.

Steps to Import Customized Feed & Speed Data in 64-bit CAMWorks Feed & Speed Library

- 1. Launch the CAMWorks Technology Database.
- 2. Click on the Feed/ Speed menu 🗮 on the left-hand side.
- 3. The Feed/Speed menu options will be displayed. Click on the first option labelled



- 4. This action will launch the CAMWorks Feed and Speed Library. Click on the command on its ribbon bar.
- A warning message will inform you that all existing data within the *Feed and Speed Library* (including customized data, if any) will be replaced on going ahead with the import process. Click **Yes** within the warning message to confirm.
- 6. As your previous *Feed and Speed Library* version was a 32-bit application with a MS Access based database engine, the current 64-bit *Feed and Speed Library* application (with a SQL database engine) will fail to find the database. A warning message will be displayed stating that no data was found and that you manually need to enter the path to the database of the previous version. Click **OK** within this message box.
- 7. The *Windows File Explorer* will be launched. Use it to navigate to the *.mdb file (MS Access database) of the previous *Feed and Speed Library* application. The default path to the 32-bit *CAMWorks Feed and Speed Library* application is:

C:\CAMWorksData\MATLIB\MATERIAL.MDB

- 8. Select the *Material.MDB* file and execute the *Open* command.
- 9. The *File Explorer* will close and the *Feeds Speeds Import* window will be displayed. This window indicates the path to the database file from which Feed & Speed data will be imported. Click on the *Import* button within this window to commence importing.

隊 Feeds Speeds Import	_		×
Previous Database Path	DB		
Select Fi	le Import	Car	ncel
Select a Database file			.::
Feeds Speeds Impo	rt Window		

Import Feeds and Speeds



10. A warning message w from the selected dat	ill prompt you confirm whether yabase. Click Yes within this mess	you are sure about importing all data sage box.
	Material Library	×
	Are you sure you want to import all data	from this database?
	<u>Y</u>	es <u>N</u> o
	Warning Message Prompt to Cor	ifirm Import Process
11. The import process w	ill commence. The user interface	e will revert to the Feed and Speed Library
application. Observe	that in the ribbon bar, the 📲 🖿	port Feeds and Speeds button command has
been temporarily repl on the volume of the 12. Once the import is co	aced with the Importing Now data present, this process will ta mplete, a message indicating su	command (in disabled state). Depending ike several minutes. ccessful import of the data will be
displayed in the statu	s bar of the user interface. The	Importing Now button will be replaced
with the Import Feed	s and Speeds button.	
Material Library		- 🗆 ×
Materials 🛉 💢 🧻 Install from Arch	ive 🕌 Feeds and Speeds 📳 Import Feeds an	d Speeds 🛛 🏟 🚱 🗐 🕬 🛛 Close
Carbon Steels Alloy Steels Stainless Steels High Temperature Alloy Steels	Ma	aterial Name:
···· Tool Steels ···· Titanium Alloys	Common Name	Material Name 6
⊕ Gray Cast Iron & Nodular ⊕ Aluminum	Unified Number	Material Name 7
···· Magnesium ⊞·· Copper / Copper Alloy ···· Plastics	ASTM Number	Material Name 8
Wood ⊞ Special Material	SAE Name	Material Name 9
	Material Name 5	Material Name 10
	Description	
	beaupton	
	Import Complete.	
13. Your 64-bit SQL base data. The next time y CAMWorks application you imported in to the	Status of Import Process indicate d database engine <i>Feed and Spee</i> ou use TechDB or retrieve inform the Feed/ Speed values applie e 64-bit SQL based database eng	ed In Status bar ad Library now contains all your customized nation from the TechDB while using the d will be based on the customized data gine Feed and Speed Library application.



ShopFloor Publisher

New - Option to copy/reference additional documents in a ShopFloor File to be Published

Purpose:

To provide the option to copy/reference additional documents in a ShopFloor file to be published

Implementation:

From **CAMWorks 2021 SP0** version onwards, you can provide additional files in any file format as a copy or file reference within the published ShopFloor file.

Use the parameters available within the newly introduced **Additional References** group box available in the **CAMWorks ShopFloor Publisher** dialog box for this purpose. Use the parameters within this group box to:

- Add a copy of one or more files of choice
- Create references to one or more files of choice
- Change any file copies or references

CAMWorks ShopFloor Publisher	3	
Publish		
Publish		
File Location	*	
C:\CAMWorksData\CAMWorks2021x64\Examples\		Add reference file ? ×
Output G-code (M3Axis-Tutorial.CTL) Change Post Processor Ouput Setup Sheets (CWViewTemplate.xsl) Change Style Sheet		File : C:\Installers\CAMWorks202Xx64-XXX\CAMWorks_ Add as : O Copy O Referance Description : Document Detailing Operations and Instructions
Additional Referances	* +	Time stamp : 2020-01-29 07:41:25
	Ţ	

Clicking on the *Add New File* button displays the *Add Reference File* dialog box. Use this dialog box to add a copy of a file of choice or provide a reference path to it in the ShopFloor file you intend to publish. Once the *Add Reference File* dialog box is closed, the file path to the selected file will be displayed in the *Additional References* list box.



The **Delete Selected File** command button will be enabled only when a file path listed in the **Additional References** list box is selected. Click on this command button if you wish to delete the selected file path reference.



ShopFloor Document Manager

New - ShopFloor Document Manager Application for Machine ShopFloor Personnel

Purpose:

To provide a zero extra cost application for Machine ShopFloor personnel that enables them all necessary information for machining parts and assemblies within a single user interface

Implementation:

ShopFloor Document Manager Application

The **ShopFloor Document Manager** is a newly introduced **<u>zero-cost</u>** standalone application now available in the CAMWorks Installer Package. It can be installed and run on any Windows system, even those on which other *CAMWorks* products are not installed.

It serves as a collaboration tool between the following sets of personnel:

- **CAM Programmers** (who use the **CAMWorks** application to generate CAM data for part models and/or assemblies to be machined)
- **Machine ShopFloor Personnel** (who execute the actual machining of the concerned part models/assemblies at the Machine ShopFloor)

Intended users of ShopFloor Document Manager Application

The Machinists working on the Machine ShopFloor are the intended users of the **ShopFloor Document Manager** application.

Installing the ShopFloor Document Manager Application

The installer for the **ShopFloor Document Manager** application is available within the **CAMWorks 2021x64** installer package within the **ShopFloorDocumentManager** sub-folder.

 > This PC > Data (C:) > Installers > CAMWorks2021x64-SP0 > ShopFloorDocumentManager

 Name
 Type

 ShopFloorDocumentManager.exe
 Application

 > Document_Manager_Installation_Guide.pdf
 Adobe Acrobat Document

ShopFloor Document Manager Installer within CAMWorks Installer Package

Current Collaboration Process followed by CAM Programmers & Machinists

- i. Once the CAM Programmer fully programs the part model/assembly using **CAMWorks** (i.e. the toolpaths have been verified and G-code has been generated), he/she generates the Setup Sheets and G-code file using associated commands in the application.
- ii. The Setups Sheets and G-code file are then shared with the Machine ShopFloor personnel.
- iii. The Setup Sheets are used for referring information on how to set up the stock, the machine, the controller, estimated machine time, part material, and the tooling used to machine the part. Optionally, the Setup Sheets may also include WIP images, Part/assembly images and/or Tool images.
- iv. The Machine ShopFloor personnel use 3rd party-software to view/edit the G-code file.
- v. After verifying the G-code, the part model/assembly is machined as per specifications.

How the Collaboration Process will work with CAMWorks + ShopFloor Document Manager Combination

The CAMWorks application contains a Publish ShopFloor functionality that allows CAM programmers to publish a ShopFloor file for the part model/ assembly to be machined.

This **ShopFloor** file (with *. *cwspflr* file extension) serves as a digital container for the following manufacturing data required by Machine ShopFloor personnel for machining part models/ assemblies:



- Data files required for CAD visualization (Part Model/Assembly view)
- CAM data
- Toolpath Simulation data
- G-code
- Setup Sheets

Such a published **ShopFloor** file can then be shared with the Machine ShopFloor personnel. The Machine ShopFloor personnel can then use the **ShopFloor Document Manager** application to view info contained in the ShopFloor file.

On launching the **ShopFloor Document Manager** application. A **File Explorer** window will prompt the user to select the ShopFloor file to be viewed. Once the file is selected, the ShopFloor Document Manager user interface will display information contained within the ShopFloor file.

⑤ CAMWorks ShopFloor Document Manager ? ×					
Save					
Part / Assembly			~		
File Name	Size	Time			
🔗 bracket.SLDPRT	16	2020-11-03 13:21:32			
G Code			*		
File Name	Size	Time			
📄 bracket.set	2 KB	2020-11-05 13:22:31			
📄 bracket.txt	9 KB	2020-11-05 13:22:31			
Setup Sheets			*		
File Name	Size	Time			
📄 bracket.SLDPRT.doc	38 KB	2020-11-05 13:22:41	1		
📄 bracket.SLDPRT.xls	38 KB	2020-11-05 13:22:41			
📄 bracket.SLDPRT.xml	88 KB	2020-11-05 13:22:41			
Additional Documents		+ ×	*		
File Name	Size	Time			
📄 AnyFile.jpg	30	2020-09-01 16:07:10			
🔗 AssetData.txt	1 KB	2019-03-19 06:32:09			
<			>		
© 2020 - Geometric Americas,	lnc.	Version: 2	021.01		

User Interface of the ShopFloor Document Manager

Value Proposition offered by Publishing & Viewing ShopFloor Files

- All the manufacturing data required by the Machine ShopFloor personnel is made available in a digital format (*. cwspflr) within a single user interface (viz. the UI of the CAMWorks ShopFloor application.)
- The association of the manufacturing data with the part model/assembly is maintained
- . No need for an expensive **SOLIDWORKS** license for viewing the part model & associated CAM data [Either the **ShopFloor Document Manager** (a freeware application) or the **CAMWorks ShopFloor** (application with cost-effective license) can be used].