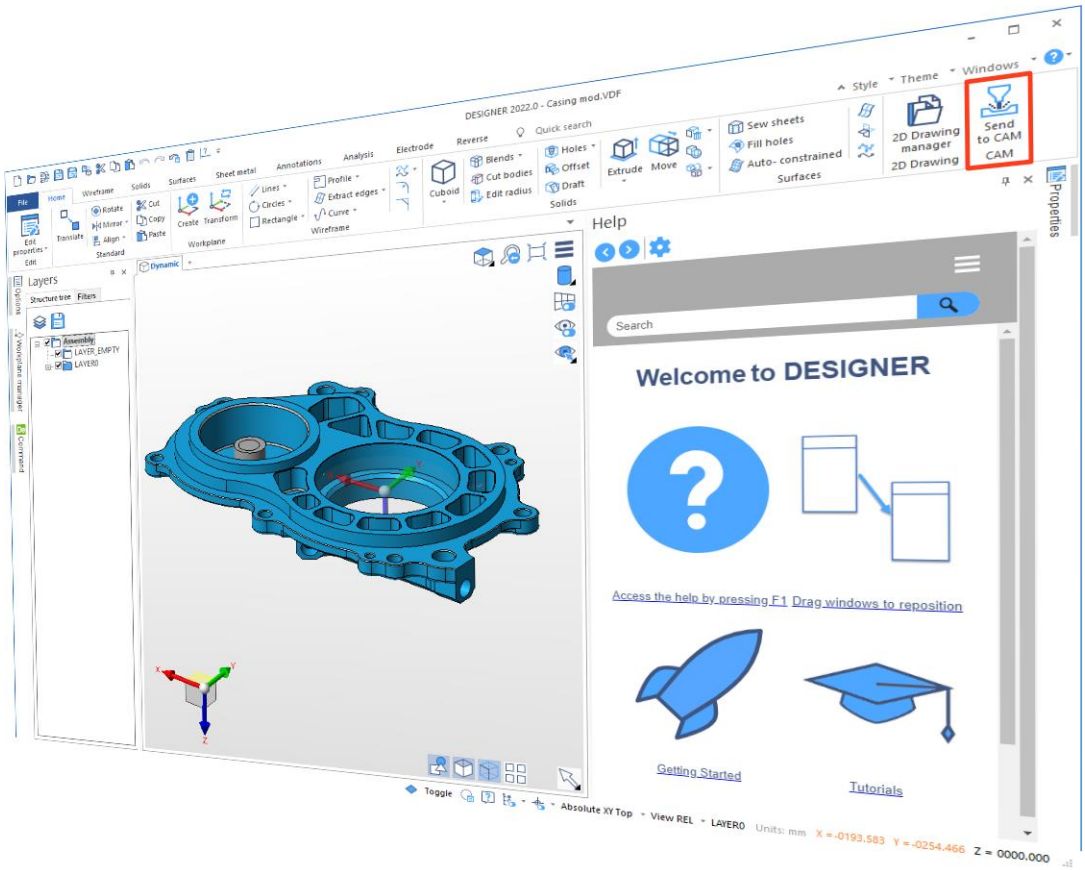
WHAT'S NEW IN EDGECAM 2022.0



This document highlights new product features and enhancements in EDGECAM 2022.0.

To run EDGECAM 2022.0, the maintenance expiry date in the license must be April 2021 or later.

17 May 2021

Contents

'What's New' Document Overview 4
Purpose of this Document and Other Sources of Information 4
Targeted Information inside EDGECAM and Other Programs 4
The Development History of EDGECAM 4
Important Information 5
Windows 7 and 8.1 Support Removed 5
EWS Retirement 5
Old AutoCAD DWG/DXF Loader retired 5
Network License Manager Upgrade 5
Manufacture Enhancements 6
EWS Retirement 6
DESIGNER Companion License 6
Export EWS models as STEP 7
SolidWorks 2021 supported 7
Solid Edge 2021 supported 7
Granite update to support for Creo V7 7
Preferences - Load SolidWorks Assembly files directly 7
Preferences - Load Inventor models as Parasolid 8
Preferences - Load SpaceClaim and ACIS models as Parasolid 9
Preferences - Load Catia models as Parasolid 9
Preferences - Option to control CAD model check period 10
Horizontal Head AC axis + B axis Table support 10
Simulate Turn/Mill with X Axis on the spindle 11
NCSIMUL Essential - Simulate Toolpaths directly from EDGECAM 11
Roughing Cycle - Improve Waveform multithreading performance 11
Roughing Cycle - Minimum Helix Diameter available for all Roughing Strategies 12
Profiling Cycle - Add Multiple Pass and Corner Type support for Dovetail Cutters 12
Profiling - Apply Offset to Protect Solid 13
Rough and Profile a hole feature with a Through Limit 13
Text Mill Cycle - Approach At Clearance option added 13

Slot Milling Cycle - Approach At Clearance available for all strategies 14
Five Axis Cycle - Calculation Based on Tool Centre Modifier 14
Standard 4-axis Simultaneous Milling license can use Advanced 5 Axis Cycle 14
5-axis Simultaneous Milling merged with Advanced 5-axis Simultaneous Milling 15
Rough Turn and Rough Profile - Change Axes Retract Order option added 16
Thread Profiling Cycle - Added option to reverse thread direction 17
Finish Turning - Option added to Reverse Side 17
Rough Waveform Turn - Smooth option added 18
Finish Groove - Break Corners can be applied to either side of groove 18
Wire EDM - Support for AC Uniqua Machine 18
Wire - Technology update for Accutex (AU-700 IA) 18
Migrate Tool - SURFCAM 2021.0 to EDGECAM 2022.0 18
Maintenance Database Report 19
New Features in Version 2021.0 20

‘What’s New’ Document Overview

Purpose of this Document and Other Sources of Information

The purpose of the document is to highlight new and changed items in the current release. Non-release specific information such as installation and licensing information, system requirements and CAD Links information can be found in the relevant document.

For help with your installation, please refer to the Installation Guide. This is available from the DVD or the Help sub-menu in the EDGECAM program group.

For help with licensing your standalone or network license, please refer to the Licensing Guide. This is available from the Help sub-menu in the EDGECAM program group, the CLS menu and the License Manager dialog.

For information on system requirements and supported CAD systems, please refer to the Installation Guide.

Targeted Information inside EDGECAM and Other Programs

In addition to this document, ‘targeted’ information on new items is available in the dialog help and user guides for other applications. This allows you to focus on new features/enhancements for a specific program or the cycle you are currently working on, for example.

Dialogs that have new functionality or where the cycle behaviour has changed have an additional ‘What’s New’ tab in the help. This explains what has been added to the dialog or changed in this release.

What’s new topic(s) have been added to help files for other programs, such as Code Wizard, Code Generator, and ToolStore etc. This only lists new functionality for that program, allowing you to focus on those items.

The Development History of EDGECAM

Additional functionality and enhancements are developed with each release of EDGECAM software. For an overview of new features and enhancements in the last release, please refer to New Features in Version 2021.0.

For a summary of new features in previous releases, please visit the History section of the EDGECAM website.

Important Information

Windows 7 and 8.1 Support Removed

Windows 7 and 8.1 are no longer supported and the EDGECAM installer will only install on Windows 10.

Live Job Reports will continue to install but we will not officially support it.

EWS Retirement

EDGECAM 2022.0 will be the last version to support EWS models.

Old AutoCAD DWG/DXF Loader retired

The old PDI based loader has been retired and removed from the Preferences dialog. The New DWG/DXF Import option has been removed and defaulted to On.

Network License Manager Upgrade

For network licensing, EDGECAM 2022.0 requires the Sentinel RMS Network License Manager version 9.7 or greater.

When EDGECAM 2022.0 connects to a network license server, if it detects an earlier version than 9.7, this message will be displayed:

‘This version of CLS is not compatible with the server you're attempting to connect to. Please upgrade the license manager on server <server name> to 9.7 or greater.’

Sentinel RMS 9.7 License Manager is included in your installation media and/or on the Software Downloads page in the Customer Portal. To install the License Manager, run setup.exe and follow the on-screen instructions. The installer will upgrade the older License Manager on the machine.

Manufacture Enhancements

EWS Retirement

EDGECAM WORKFLOW SOLIDS, EWS, has been the solid modelling solution for EDGECAM customers seeking a simple way to create fixtures, stock models, and accessory 3D representations of elements of the manufacturing process.

Hexagon's DESIGNER, the next generation CAD application for Manufacturing, is a whole new level in functionality, ease of use, and power.

With DESIGNER available to all EDGECAM customers on maintenance, we will retire EWS from EDGECAM.

EDGECAM 2022.0 will be the last version to support EWS. To open existing PPF with EWS models, a module license will be required: **ENEWS-M "EDGECAM EWS Module**". This license will be free of charge, and available to existing EDGECAM users upon request. It will not be available to new customers. Customers trying to run EWS on 2022.0 without the license will receive a warning to contact us and a 10 days grace period.

This version introduces the **Save EWS as STEP** command which can be run from inside EDGECAM, or via the command line batch file.

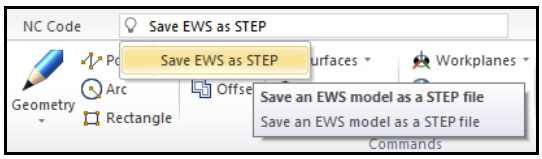
Future versions of EDGECAM will not support EWS. They will warn the user that the selected PPF cannot be opened in this version and suggest opening it in 2022.0 for manual treatment.

DESIGNER Companion License

With the retirement of EWS in the next EDGECAM version, we have allowed DESIGNER Standard to be run in 2022.0 as the new CAD solution. As with EWS, DESIGNER will run once you agree to the term and conditions similar to those required for EWS:

- DESIGNER Companion will be added to all customer license configurations that are in maintenance when we release EDGECAM 2022.0.
- The DESIGNER Companion licenses will be free of charge timed licenses that will expire on the Maintenance expiry date of the EDGECAM licenses.
- When the maintenance on the EDGECAM licenses is renewed, the DESIGNER Companion licenses will be extended to the new maintenance expiry date.
- Customers will need to install the new license to get the DESIGNER Companion license(s).
- For network licenses, a DESIGNER Companion license will be added for each EDGECAM system license in the configuration, for example if there are 2 x EDGECAM Standard Milling + 3 EDGECAM Advanced Turning licenses on a network license, we will add 5 DESIGNER Companion licenses.
- For network license configurations, the user must configure a prerequisite EDGECAM system license before they can select the DESIGNER Companion license. When DESIGNER is run, the EDGECAM system license is taken from the license pool with a DESIGNER Companion license.
- Compared with EDGECAM Designer, the DESIGNER system license and DESIGNER Companion license are the equivalent of the EDGECAM Standard Designer license + Analysis +2D Drawing.

Export EWS models as STEP



We now have the ability to export the EWS model in the EDGECAM part file to a STEP model. This will allow our customers to take that STEP file into DESIGNER for modification or annotation.

The command is not directly on the interface, but can be started via the Quick Search bar by typing **Save EWS as STEP**. We have also supplied a PCI-JS script that will ask for a source and destination folders and export STEP models for all the EDGECAM part files in the source folder.

SolidWorks 2021 supported

EDGECAM now supports SolidWorks 2021.

Solid Edge 2021 supported

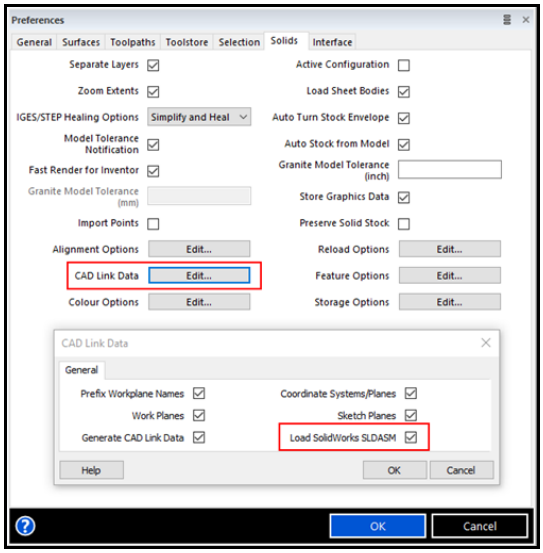
EDGECAM now supports Solid Edge 2021.

Granite update to support for Creo V7

EDGECAM has been updated to support Creo v7 parts for the Solid Machinist for Granite license.

Although this license is no longer on the price list, we update the version for our existing customers.

Preferences - Load SolidWorks Assembly files directly



EDGECAM can now load SLDASM files.

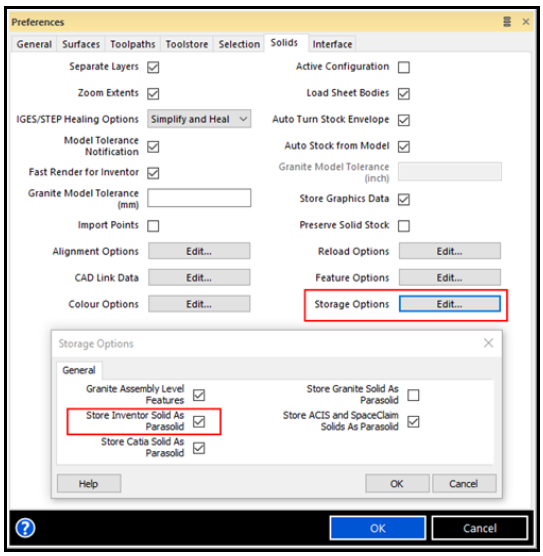
Previously, the CADLink button in SolidWorks had to not only create the XML data file for Features, but also create a X_B of the assembly and use that to start EDGECAM.

CADLinks

A new **Load SolidWorks SLDASM** preference has been added for this release which controls how CADLinks passes the model to EDGECAM:

- Checked- CADLinks will load the SLDASM file into EDGECAM.
- Unchecked - CADLinks will create a X_B file of the models in the assembly and load the X_B file into EDGECAM (Current behaviour).

Preferences - Load Inventor models as Parasolid



As part of the alignment of EDGECAM with EDGECAM Designer we are moving towards all solid models being loaded as Parasolid models. This gives us a number of advantages depending on the model type:

- No need to install Autodesk Inventor view.
- Designer and EDGECAM will use the same loader (Solidlink).
- CADLinks will continue to work.
- Face triangulation is more robust in Parasolid and can improve toolpaths.

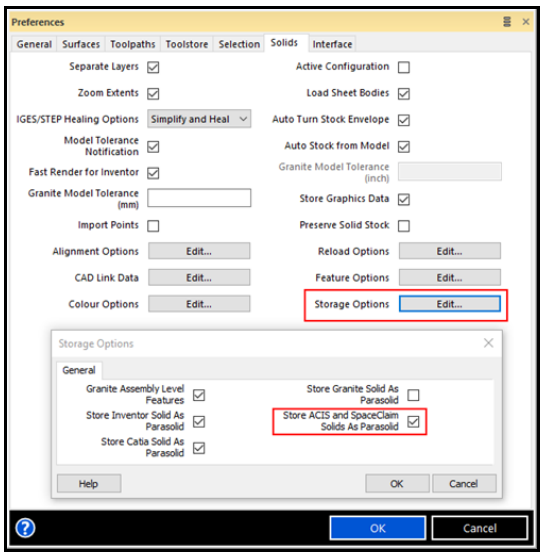
A new **Store Inventor Solid As Parasolid** option has been added to the Storage Options section in the Solids tab of the Preferences dialog.

Existing part files containing ipt will continue to be stored as Inventor models and, when updated, will remain as Inventor models. There will be no conversion for existing parts.

When the preference is selected, the model will be coloured white (parasolid) rather than goldenrod when loaded if the **Ignore Model Values** colour option is enabled.

Note: Assembly files (.iam) need to be Inventor 2008 or later.

Preferences - Load SpaceClaim and ACIS models as Parasolid



As part of the alignment of EDGECAM with EDGECAM Designer we are moving towards all solid models being loaded as Parasolid models. This gives us a number of advantages depending on the model type:

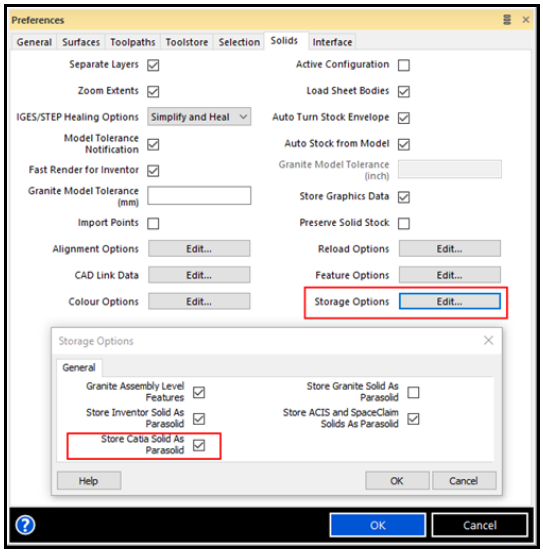
- Designer and EDGECAM will use the same loader (Solidlink).
- CADLinks will continue to work.
- Face triangulation is more robust in Parasolid and can improve toolpaths.

A new **Store ACIS and SpaceClaim Solids As Parasolid** option has been added to the Storage Options section in the Solids tab of the Preferences dialog.

Existing part files containing the native file type (.sat, sab and scdoc) will continue to be stored as native models and, when updated, will remain as native models. There will be no conversion for existing parts.

When the preference is selected, the model will be coloured white (parasolid) rather than slate blue when loaded if the **Ignore Model Values** colour option is enabled.

Preferences - Load Catia models as Parasolid



As part of the alignment of EDGECAM with EDGECAM Designer we are moving towards all solid models being loaded as Parasolid models. This gives us a number of advantages depending on the model type:

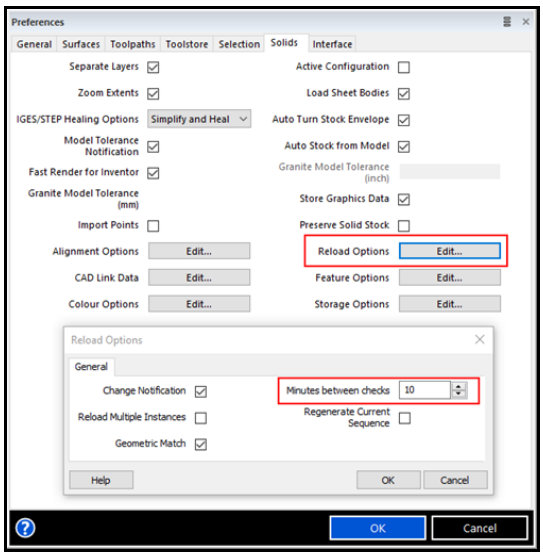
- Designer and EDGECAM will use the same loader (Solidlink).
- CADLinks will continue to work.
- Face triangulation is more robust in Parasolid and can improve toolpaths.

A new **Store Catia Solid As Parasolid** option has been added to the Storage Options section in the Solids tab of the Preferences dialog.

Existing part files containing CATPart and CATProduct will continue to store the native model and, when updated, will remain as the native model. There will be no conversion for existing parts.

When the preference is selected, the model will be coloured white (parasolid) rather than porcelain blue when loaded if the **Ignore Model Values** colour option is enabled.

Preferences - Option to control CAD model check period

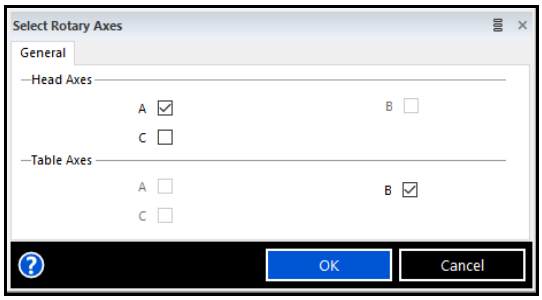


For this release, we have added a new preference to control the number of minutes between checking whether the CAD model has been updated. This can affect network performance when several users are using a network drive.

The new **Minutes between checks** option has been added to the Reload Options section in the Solids tab of the Preferences dialog.

The default is 10 minutes. If you enter 0, the check will be performed every couple of seconds.

Horizontal Head AC axis + B axis Table support



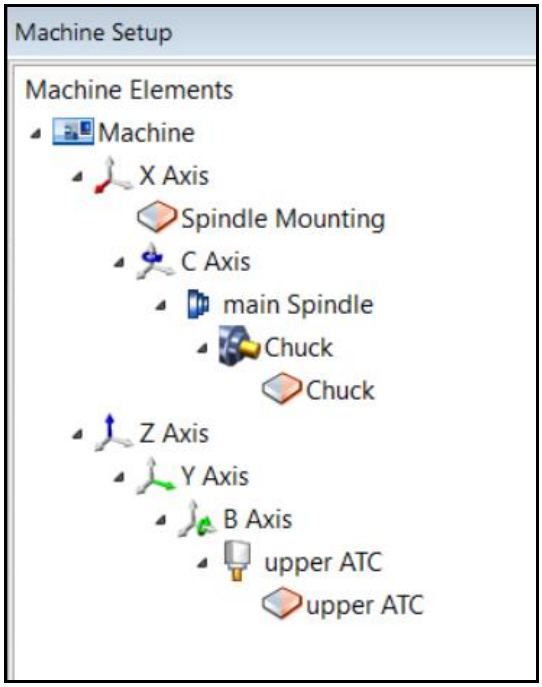
EDGECAM now supports a Horizontal Machine tool with A and C rotary axes in the head and a B axis Table.

You can control 5 axes simultaneously and we have added a command to select which two of the 3 rotary axes are active:

Machining Tab > Miscellaneous > Select Rotary Axes

This gives you the ability to decide which rotary axes to use when indexing or performing a 5 axis cycle. It also supports Rotary mode when the tool is Axial or Radial to the B Axis.

Simulate Turn/Mill with X Axis on the spindle

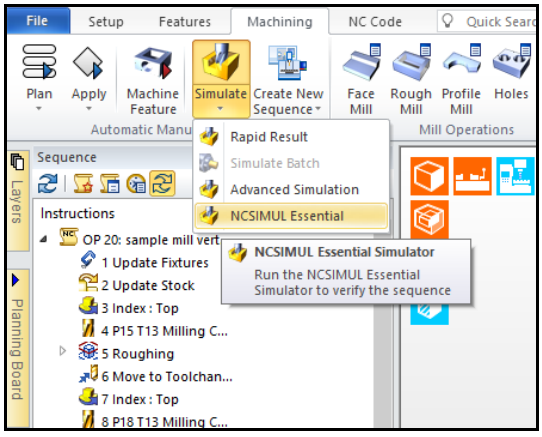


Normally a Turn/Mill CYB machine would have C in the spindle and XYZB axes in the Head.

We have added support for XC in the spindle and YZB in the head.

This is typically seen on vertical Mills with Turning capability. For example, the Mazak Integrex e-1060 series.

NCSIMUL Essential - Simulate Toolpaths directly from EDGECAM



We are introducing **NCSIMUL Essential** as a new option to simulate EDGECAM toolpaths (not NC code).

This is the first stage of the integration and will be limited to 3 axis Milling at this stage. We do not expect every 3 axis milling function to be fully supported in this release. We will, however, continue development and issue updates via software updates.

NCSIMUL Essential is included in all Milling systems. It will pass the Part model, Stock, Fixtures, Tools and Toolpath to NCSIMUL for simulation.

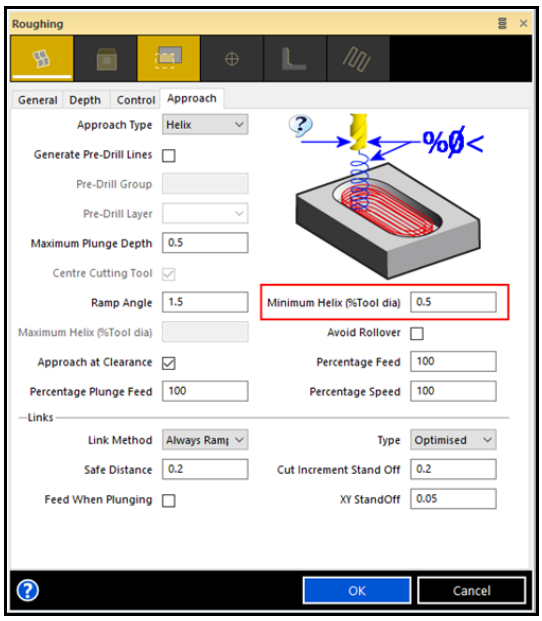
NCSIMUL Essential is started by selecting the option under **Simulate** on the Manufacturing Tab.

Note: NCSIMUL will be part of the EDGECAM installation and automatically installed.

Roughing Cycle - Improve Waveform multithreading performance

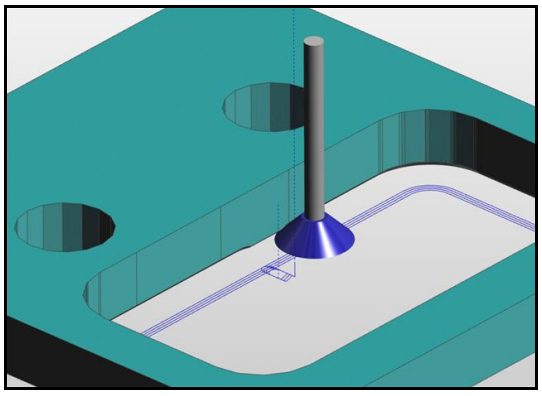
Roughing Waveform (and other strategies) have benefited from improved performance on processors with multiple cores. In particular 8 and above cores. The improvement seen can, in some cases, be 2 to 3 times faster. This is dependent on the geometry of the model and the number of available cores.

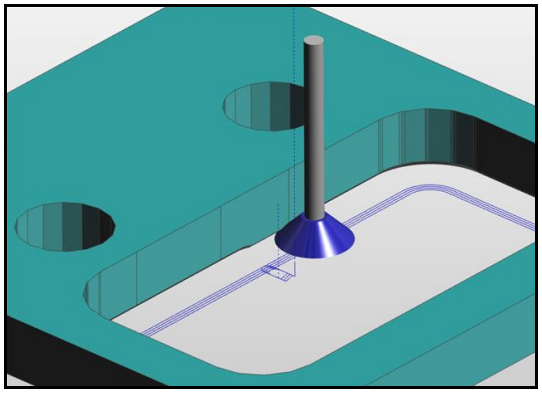
Roughing Cycle - Minimum Helix Diameter available for all Roughing Strategies



In a previous release, **Minimum Helix (%Tool dia)** was enabled for the Roughing Waveform cycle. This functionality has now been extended to the other Strategies.

Profiling Cycle - Add Multiple Pass and Corner Type support for Dovetail Cutters



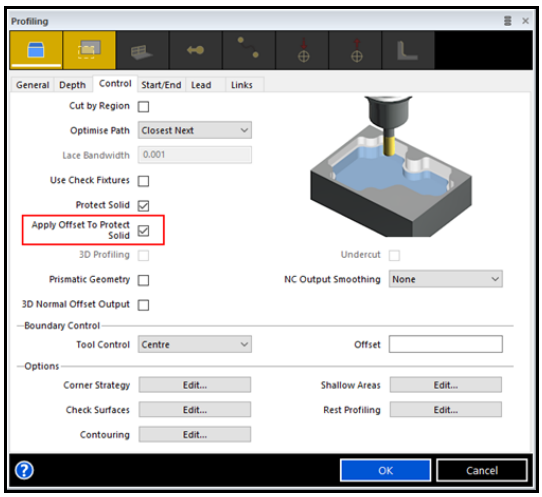


The Profiling Cycle has been enhanced to support **Multiple Passes** when using a Dovetail Cutter on Wireframe and solid edges. This option means that several passes can be taken to reduce cutting forces.

We have also made **Corner Type** available. This is useful on external corners where you want to maintain a sharp corner. Use Sharp or High Speed.

On the Toolchange dialog, the **Undercut Distance** modifier was in the Back Bore Data button. This has been relocated next to Reach on the second tab to be more accessible

Profiling - Apply Offset to Protect Solid



For this release, a new **Apply Offset to Protect Solid** modifier has been added to the Control tab.

When checked, the cycle Offset will also be applied to the solid making the solid larger or smaller by the offset value. When unchecked, the solid remains the original size.

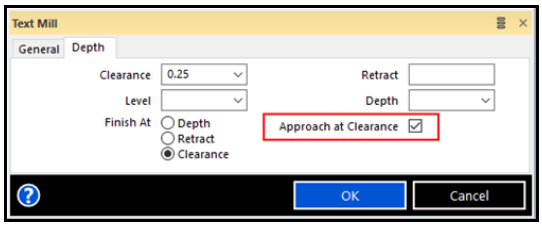
This can be useful when, for example, machining an electrode where you do want a negative offset in the toolpath to check against a solid also reduced by the same offset.

Rough and Profile a hole feature with a Through Limit

When Roughing or Profiling a **Limited Through Hole Feature**, the cycle will now allow you to cut below the Feature depth. However, it will clamp the depth to the Through Limit to avoid accidentally cutting into the other side of the part.

The user will see the message, "The depth entered is deeper than the Through Limit of the Hole Feature. Depth clamped to 'Through Limit' distance."

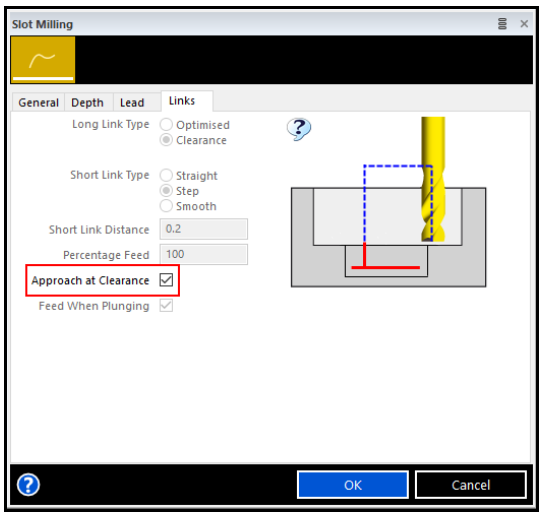
Text Mill Cycle - Approach At Clearance option added



For consistency, the **Approach At Clearance** modifier has been added to the Depth tab of the Text Mill Cycle.

Typically, a cycle would finish 'at clearance' at the end of the cycle, and you would want the next cycle to approach at clearance height. In some cases, you may want to finish at depth (by setting the 'Finish At' option on the Depth tab accordingly) and the following cycle to approach at that depth (not at clearance height). To do this, you need to ensure the 'Approach at Clearance' option for the following cycle is unchecked.

Slot Milling Cycle - Approach At Clearance available for all strategies

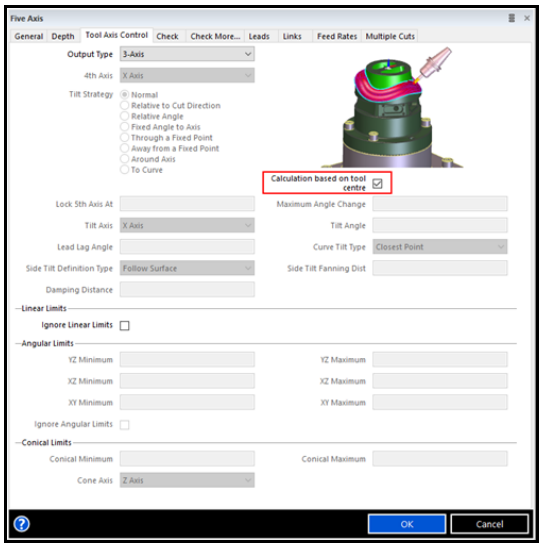


The Approach At Clearance modifier on the Slot Milling cycle is now available for all strategies.

Typically, a cycle would finish 'at clearance' at the end of the cycle, and you would want the next cycle to approach at clearance height.

In some cases, you may want to finish at depth (by setting the **Finish At** option on the Depth tab accordingly) and the following cycle to approach at that depth (not at clearance height). To do this, you need to ensure that the **Approach at Clearance** option for the following cycle is unchecked.

Five Axis Cycle - Calculation Based on Tool Centre Modifier



A new **Calculation based on tool centre** check box has been added to the Tool Axis Control tab. This option is already available in the Advanced 5 Axis cycle.

This option can be used when a bi-tangential point for parallel to surface or morph between 2 surfaces is being used.

Standard 4-axis Simultaneous Milling license can use Advanced 5 Axis Cycle

Customers with the Standard 4-axis Simultaneous Milling license now have access to the Advanced 5 Axis Cycle which will restrict the **Output format** to 3 or 4 axis.

The pre-requisite for Standard 4-Axis Simultaneous Milling now includes Standard Mill/Turn/Production.

5-axis Simultaneous Milling merged with Advanced 5-axis Simultaneous Milling

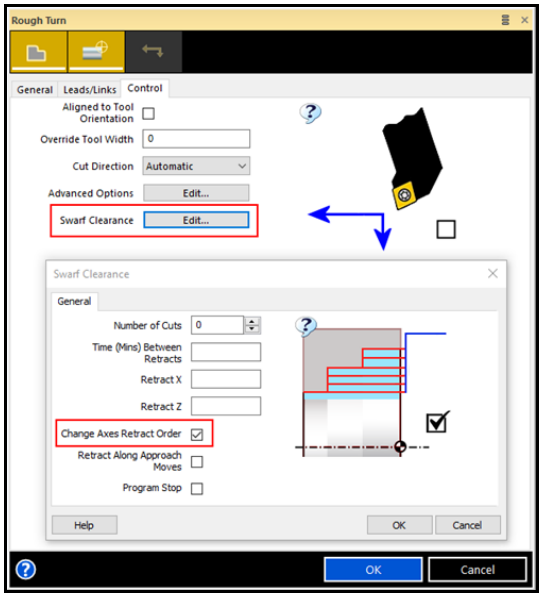
The 5-axis Simultaneous Milling (EN050-S) license will be merged into the Advanced 5-axis Simultaneous Milling (EN050-A) license to form a single license.

Customers on EN050-S will transfer to EN050-A which will have access to the 3 to 5 Axis, Five Axis and Advanced 5 Axis cycles:

- Customers with just EN050-S gain the Advanced Five Axis cycle.
- Customers with just EN050-A gain the Five Axis cycle (JS macro support).

Note: Ultimate 5-Axis Simultaneous Milling (EN050-U) remains unchanged.

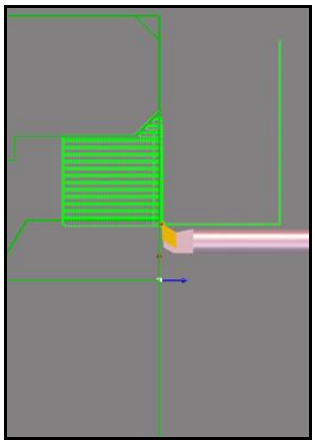
Rough Turn and Rough Profile - Change Axes Retract Order option added



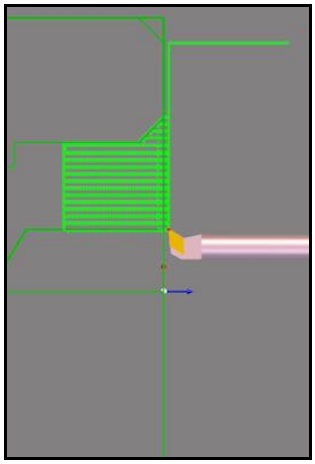
For this release, a **Change Axes Retract Order** option has been added to the Swarf Clearance dialog on the Control tab of the Rough Turn and Rough Profile cycles. This option gives the user more control over the end result of the toolpath.

When checked, the tool will switch the order of retraction. If it is originally retracting in Z then in X, it will apply X first and then Z.

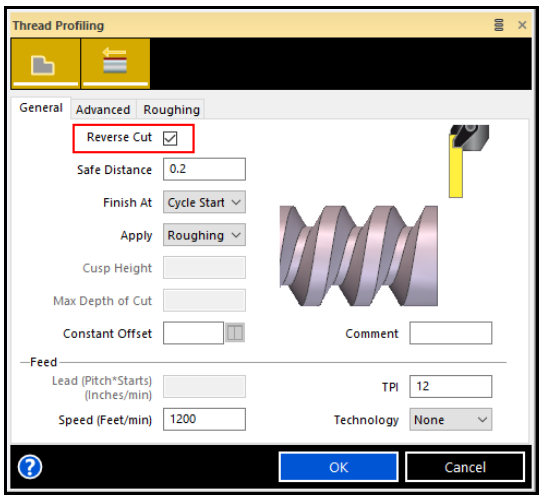
Option unchecked:



Option checked:

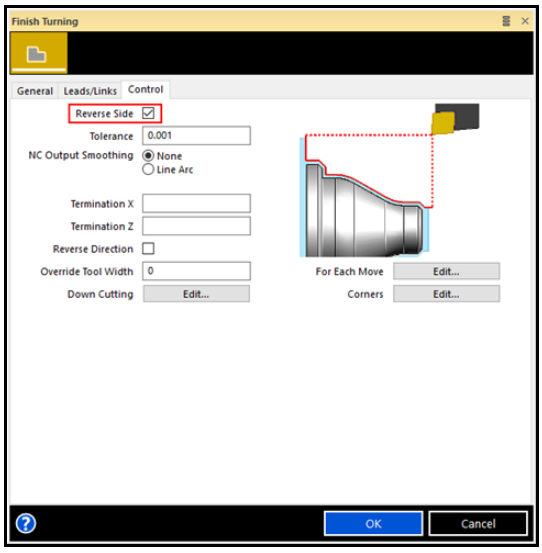


Thread Profiling Cycle - Added option to reverse thread direction



For this release, we have improved the Thread Profiling cycle by adding an option to reverse the Cut direction. This allows a Strategy or Script to control the direction without pick information.

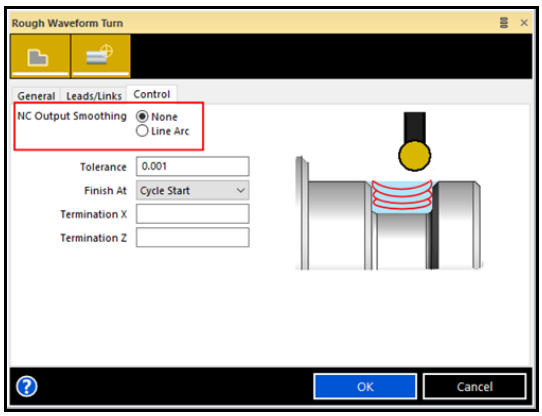
Finish Turning - Option added to Reverse Side



When Finish turning with a groove tool, and the only geometry selected is a single line parallel to the tool axis, the wrong side can be selected. We have introduced a new modifier to reverse the side if this situation is encountered.

A **Reverse Side** option has been added to the Control tab which, when checked, switches the offset to the opposite side of the selected element.

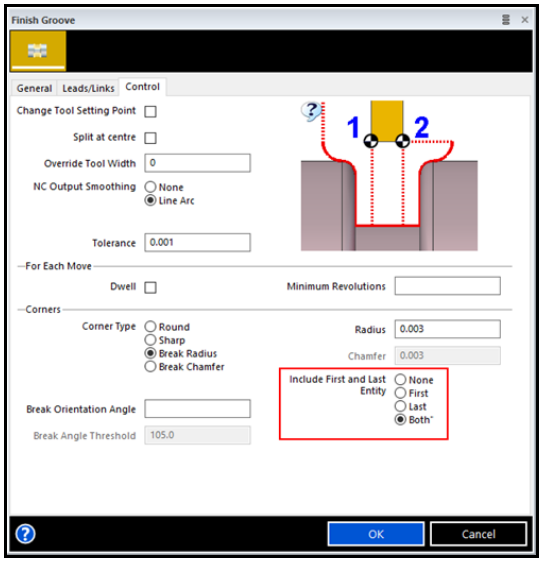
Rough Waveform Turn - Smooth option added



The Rough Waveform Turning cycle has been enhanced by adding a Smooth option to convert the line segments into fitted arcs. This can reduce the NC code by 75%.

A new **NC Output Smoothing** option has been added to the Control tab of the Rough Waveform Turn cycle which allows you to select **Line Arc** smoothing, if required.

Finish Groove - Break Corners can be applied to either side of groove



The Finish Groove cycle has been enhanced to allow the break corner to be applied to either the first, last or both sides of the groove.

Include First and Last Entity on the Control tab of the Finish Groove cycle which was previously a checkbox now allows you to select the **First**, **Last** or **Both** sides of the groove.

Wire EDM - Support for AC Uniqua Machine

A new post processor is available for the AC Uniqua machine.

Wire - Technology update for Accutex (AU-700 IA)

The technology data for the Accutex (AU-700IA) has been updated.

Migrate Tool - SURFCAM 2021.0 to EDGECAM 2022.0

The Migrate Tool has been updated to transfer SURFCAM 2020.0 onwards to EDGECAM 2022.0.

When EDGECAM 2022.0 is first started, it looks for a previous EDGECAM installation and offers to migrate the settings, post processors, scripts etc. If no previous EDGECAM is found it will check to see if there is a SURFCAM 2020.0 or later and offer to migrate it to EDGECAM 2022.0.

Maintenance Database Report

For a full list of maintenance items resolved in EDGECAM 2022.0, please refer to the Maintenance Database Report.

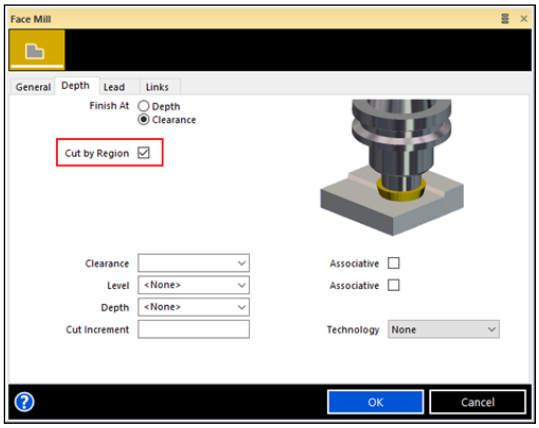
New Features in Version 2021.0

Below is an overview of new features and enhancements in the last release.

For a summary of new features in previous releases, please visit the History section of the EDGECAM website.

Manufacture Enhancements

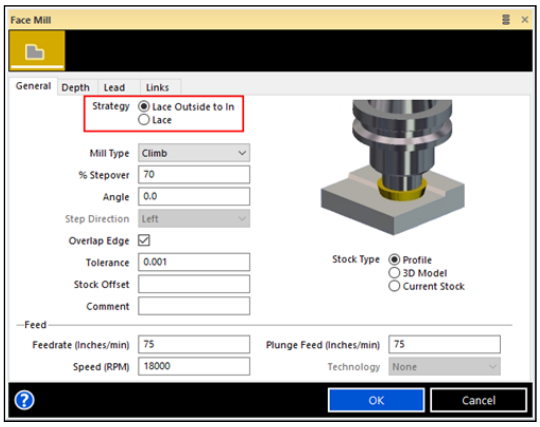
Face Mill - Cut by Region added



Because Face Milling can recognise the current stock, the cycle can be used to machine the top area of different bosses. However, previously, the cuts had no proper order and the tool could jump around between the different areas causing a lot of unnecessary moves.

For this release, a **Cut by Region** option has been added to the Depth tab in the Face Mill cycle enabling the cycle to machine all the passes of each boss before moving into the next one.

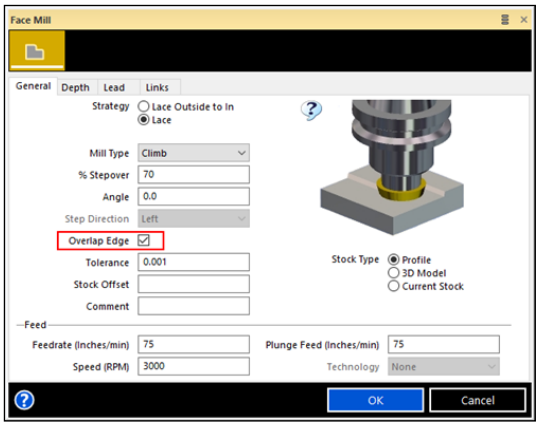
Face Mill - Outside to In Strategy added



For this release, a **Strategy** option has been added to the General tab in the Face Mill cycle enabling you to choose between:

- **Lace Outside to in** - Allows the tool to cut from the outside edges in towards the middle of the part.
- **Lace** - Parallel lace pattern as used in previous versions.

Face Mill - Overlap Edge added

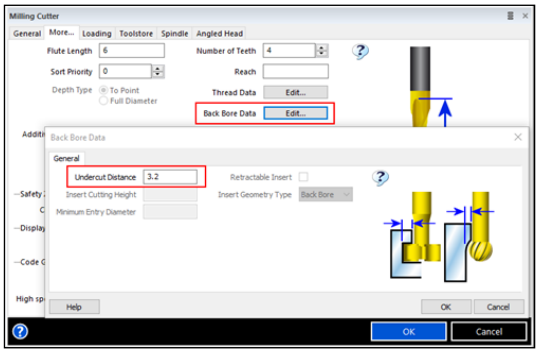


Previously, a stock offset was used to ensure that the Face Mill cycle tool cut over the stock edge but this also meant that the length of cut increased.

For this release, an **Overlap Edge** option has been added to the General tab in the Face Mill cycle which adjusts the last pass to overlap the far edge by redistributing the passes without increasing the cycle time or exceeding the stepover.

The option is checked by default, and the previous default **Stock Offset** values (0.5 mm or 0.2 in) have been removed to avoid the unnecessary increase in the length of cut.

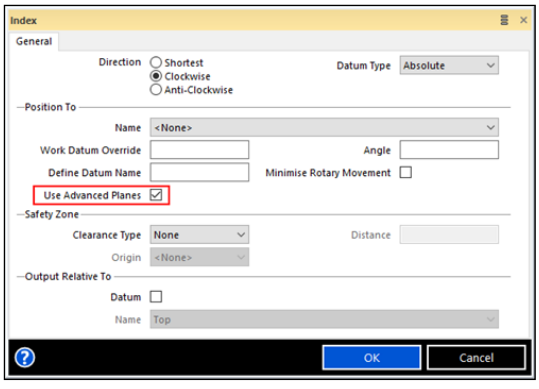
Profiling - Dovetail tools supported



For this release, we have added support for Dovetail tools in the Profiling cycle:

- An **Undercut Distance** must be set in the **Back Bore Data** section of the Milling Cutter dialog (More... tab) in the same way as it would be for a T-Slot or a Lollipop tool.
- In the Profiling dialog, **Pick Solid Faces** on the General tab and **Undercut** on the Control tab must be selected.
- You can use the Solid or a Face Feature and work with boundaries.
- You will need to pick the face that is to be machined.

Index - Support Advanced Plane transformations

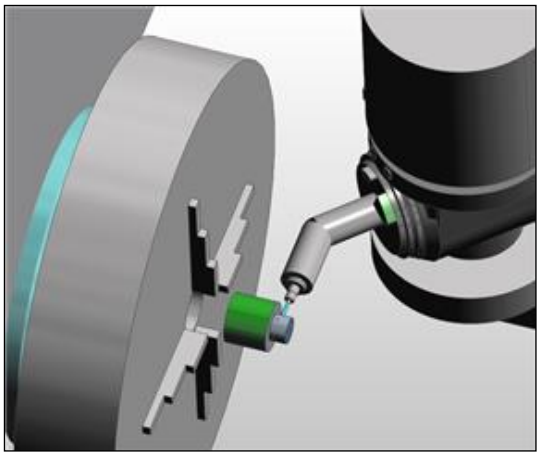


When indexing to a Workplane that would normally require it to be aligned to the machine axes, EDGECAM will leave it in its current orientation and moves and cycles will be relative to that plane.

A new **Use Advanced Planes** option has been added to the General tab of the Index dialog which, when checked, will not align the Workplane to the machine axes and instead set up a system variable for the angular difference called **WPROTATION**. A value of 0 means that the Workplane was already aligned or has been aligned and there is, therefore, no need to output a rotation in the NC file. This has the following benefits:

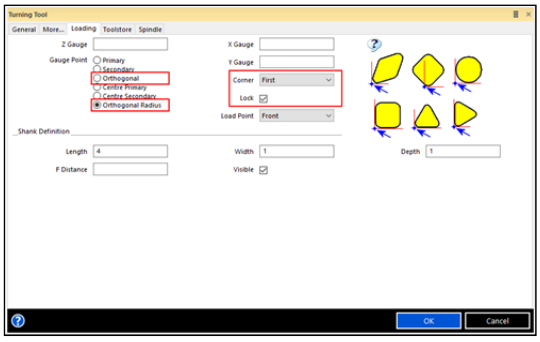
- Where parts need to be probed for position and angle, the Controller offset can be easily updated either manually or using a probe routine.
- Program relevant to drawing.
- Easier to read NC.

Angled Heads available on MTM machines



Previously, Angled Heads have only been available for Milling templates. This has now been extended for MTM based templates.

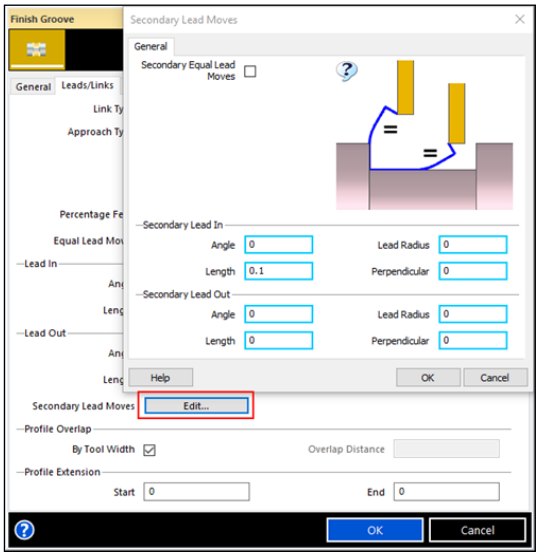
Turning / Dynamic Gauge Point for angled tools



When rotating the insert (and shank), the system will automatically detect the gauge point but, occasionally, the user may want a different corner to be used. Therefore, to control the gauge point, additional controls have been added for this release:

- **Corner and Lock** modifiers have been added to the Loading tab of the Turning Tool dialog which are available when using the **Orthogonal** and **Orthogonal Radius** gauge point options.
- The new modifiers will be available for all types of inserts, in all orientations.

Finish Groove - Secondary Lead In/Lead Out added

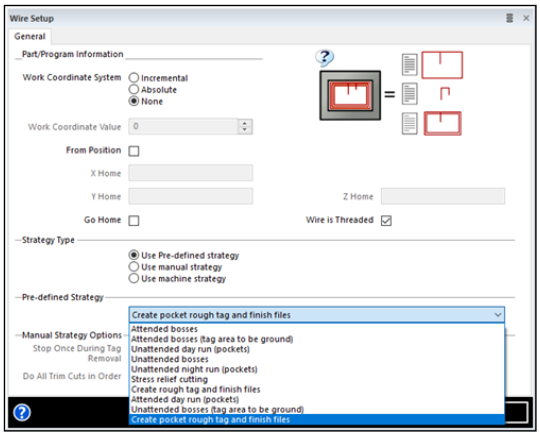


Secondary Lead Moves are now available on the Lead/Links tab of the Finish Groove cycle.

Previously, when using the Finish Groove cycle, there was no way to control the lead in and lead out moves for the second cut:

- To allow more control over the way the tool goes around the second cut, we have now added **Secondary Lead In** and **Secondary Lead Out** options.
- The original **Lead In** and **Lead Out** options continue to control the first cut of the cycle.
- As most users will expect the secondary to be the same as the first, we have made all the secondary modifiers follow the first eliminating the need to change both values. This is achieved by using formulas in the dialog.

Wire Setup - 'Create pocket rough tag and finish files' pre-defined strategy added



A **Create pocket rough tag and finish files** pre-defined strategy has been added to the **General** tab of the Wire Setup dialog.

Four separate NC files are produced when this strategy is selected:

- Pocketing file.
- Roughing pass file leaving a tag(s).
- Tag pass file to remove tag(s).
- Finishing file containing the finishing passes.

EDGECAM Inspect improvements

As part of the ongoing improvements to EDGECAM Inspect, a number of enhancements have been implemented:

- **Edit multiple features**

  The Edit Properties dialog has been re-introduced. This dialog allows the user to overwrite specific properties for all selected features. The dialog is displayed when you pick multiple features.

- **Change Layer and Colour using the Edit command**

  With this improvement, you can change the Layer and Colour of the selected features when using the Edit properties dialog.

- **Projection dialog**

  The Inspect Linear and Rectangular Array dialogs have been re-introduced to EDGECAM Inspect.

- **PCI Recording for Inspect features and function**

  Recording of PCI-JS has been implemented in EDGECAM Inspect. and PCI-JS can now be recorded for all feature types and Inspect commands

- **Move Multiple Points and Copy**

  The EDGECAM Inspect Move command has been improved and you can now move features using different methods:

  - If you pick the feature from the screen, you will be prompted to select new location as previously.
  - If you pick a feature or multiple features from the Feature browser, a dialog will be displayed with three **Action** options (**Translate**, **Move To** and **Offset**) and a Copy option which will create a copy of the feature using the selected action.

- **Constructed Conical**

  A new Constructed feature is available which can be constructed from circular features (Cylinder, Cone, Hole). With this feature, the user can evaluate the Conicity/Cylindricity of multiple faces.

- **Support of misaligned arcs**

  Previously, the feature engine for Inspect features did not support misaligned arcs. This issue has now been resolved and arcs can now be drawn in different planes from the one to which the feature is assigned.

General Performance improvements

We have been looking at a number of areas where EDGECAM slowed down when working with very large part files and have addressed a number of these issues:

- **Deleting geometry slow when sequence has hundreds of instructions**

  The problem was that the Sequence browser was being refreshed whether or not the geometry was used in a cycle. There is now no delay.

- **Layers can take a long time to turn on/off**

  This would be seen in parts with a lot of instructions and features and the browser being updated when not necessary. A customer part took over a minute to respond and now it is just a second.

- **Part slow to start regeneration**

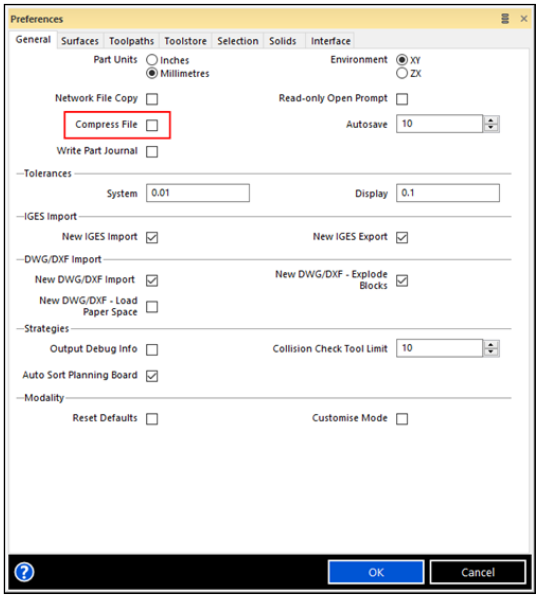
  Parts with complex components were generating an STL model for the Simulator every time. The STL is now cached and stored in the PPF saving a substantial amount of time.

Rough Waveform Performance improvements

The Rough Waveform cycle has been optimised by improving parallel processing. On average, the cycle is twice as fast but can be up to four times faster.

Note: Multiple cores are required to benefit from this enhancement.

Preferences - Option to compress STL models in Simulator



Simulator now stores the STL models generated by the stock and part solids during the simulation within the PPF file.

A new **Compress File** option has been added to the General tab of the Preferences dialog:

- Check this to compress the STL models and reduce the size of the PPF file although the compression process will take extra time.
- Uncheck if disk space is not a concern or the extra time taken would be an issue.

Simulator Enhancements

Simulator models retained for better performance

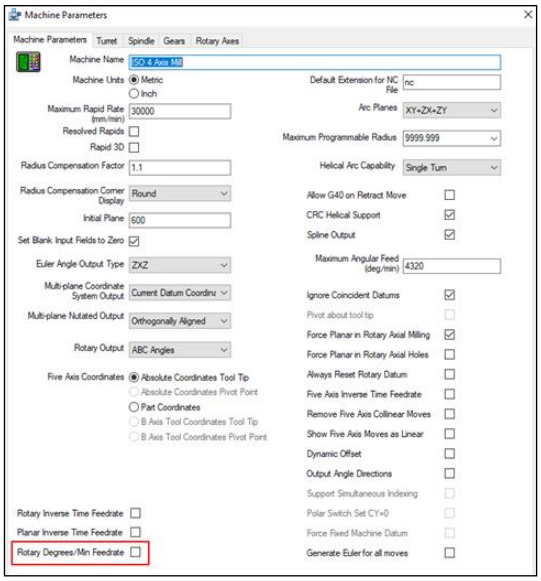
Simulator now stores the STL models generated by the stock and part solids during the simulation within the PPF file.

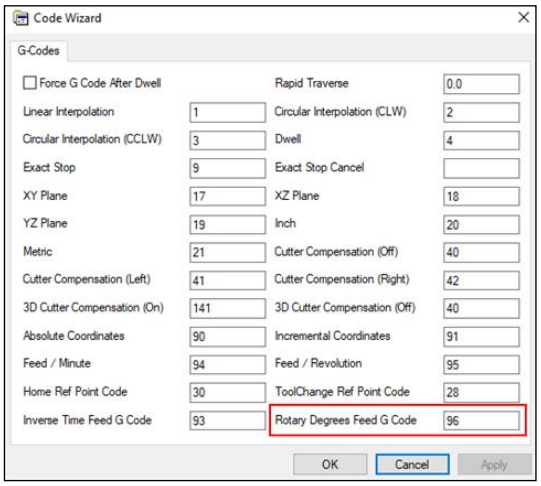
This means that when the file is opened again in Simulator, either within the same session or in a new session, it will be much faster, even when the file is used on another computer or by another user.

In previous versions, Simulator would only open faster within the same session.

Code Wizard Enhancements

Rotary Degrees/Min Feedrate



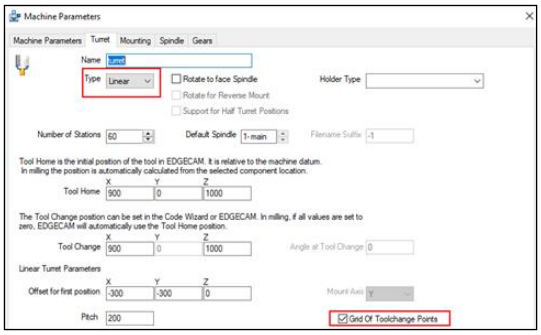


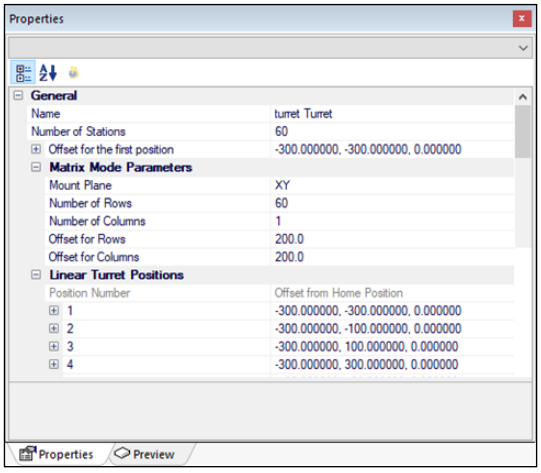
A **Rotary Degrees/Min Feedrate** option has been added to the **Machine Parameters** tab of the **Machine Parameters** dialog.

This modifier, available for any machine type, will change the FEED system variable value in Rotary moves by converting the feed values into degrees per minute.

It is possible to change the value of the **Rotary Degrees Feed G Code** on the **G-Codes** tab of the **NC Style**, **G-Codes and Modality** section.

Grid of toolchange points





For linear mounted tools, EDGECAM supports a single line with a single pitch value. Some machine tools have variable pitch and also multiple axes to mount the tools.

A **Grid of Toolchange Points** option has been added to the **Turret** tab of the **Machine Parameters** dialog that will allow the construction of turrets with toolchange points distributed in a grid:

- When this option is checked, additional modifiers will be available under the Turret Properties which will provide the user with a default structure for a quick grid style distribution of the toolchange points.
- The toolchange points can be configured and override any configuration previously set. These coordinates specify the distances around each toolchange point, allowing the user to distribute toolchange points within the grid.

Note: These coordinates are measured from the Turret Home position.