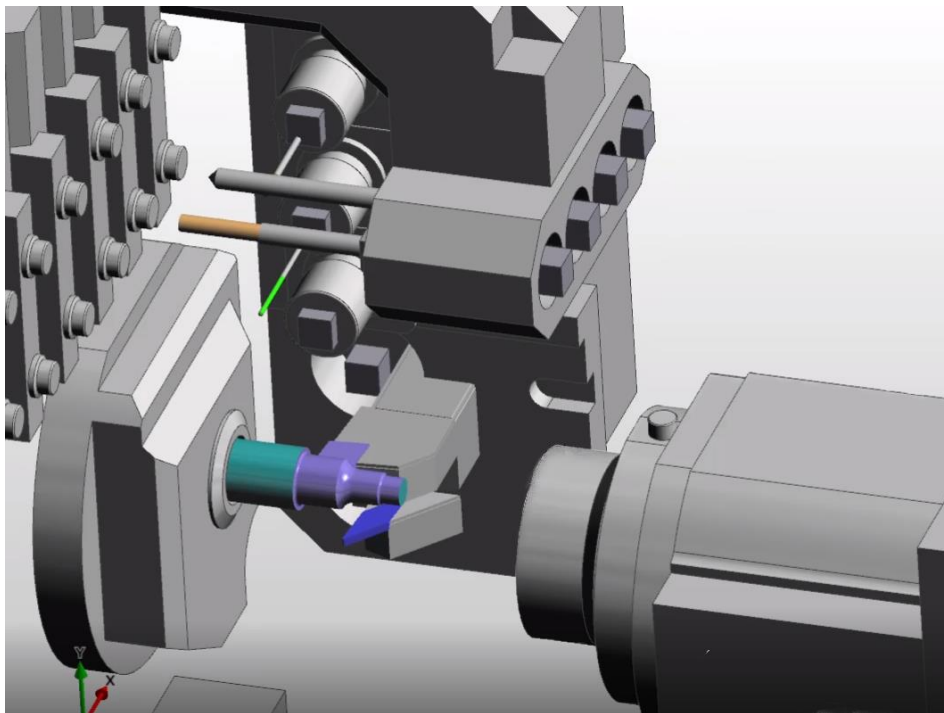# WHAT'S NEW IN EDGECAM 2021.0

This document highlights new product features and enhancements in EDGECAM 2021.0.

To run EDGECAM 2021.0, the maintenance expiry date in the license must be March 2020 or later.

25/03/2020

Contents

'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 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 2020.1.

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

Microsoft ended extended support for Windows 7 in January 2020.

Hexagon Production Software will officially support Windows 7 and Windows 8.1 for the supported life of the 2020 product releases.

As our key component suppliers start to withdraw support for Windows 7 and Windows 8.1, our 2021.0 release will not officially support these platforms.

Please consider updating your operating system to ensure that you are running your software on a tested and supported platform.

We would recommend updating to Windows 10 Build 1809 or later.

DVD Image not supplied

The main EDGECAM Installation zip file will no longer be referred to as the 'DVD Image'. We are no longer supplying DVDs and the zip file size will exceed the DVD storage limit. For example, EC2020.1_English_DVD_Image.zip will now be called EC2021.0_English_Installation.zip.

Designer installation

EDGECAM will now install Designer as part of the standard installation.

PDI based 5 axis cycle retirement

The PDI based 5 axis cycles have not been available on the standard interface, have been off the price list and unsupported for a number of releases. They will be removed from the EDGECAM 2021.1 installation:

- Five Axis Areaclear
- Five Axis Drill
- Five Axis Slot

Manufacture Enhancements

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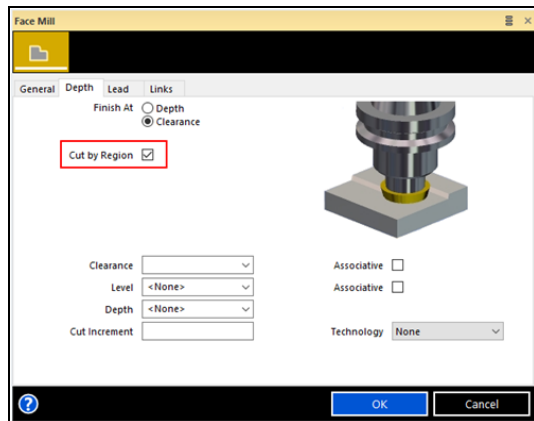
Because Face Milling can recognise the current stock, the cycle can be used to machine the top area of different bosses. However, previously, the cuts had no proper order and the tool could jump around between the different areas causing a lot of unnecessary moves.

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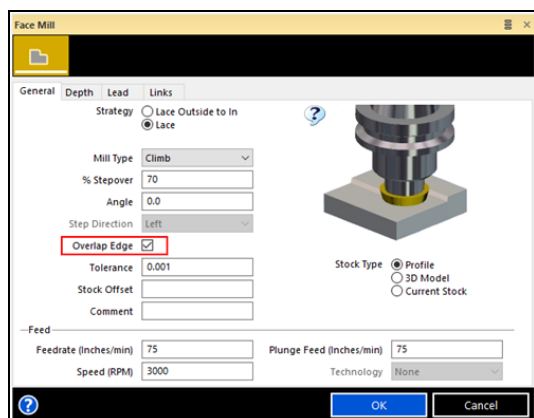
Face Mill - Outside to In Strategy added
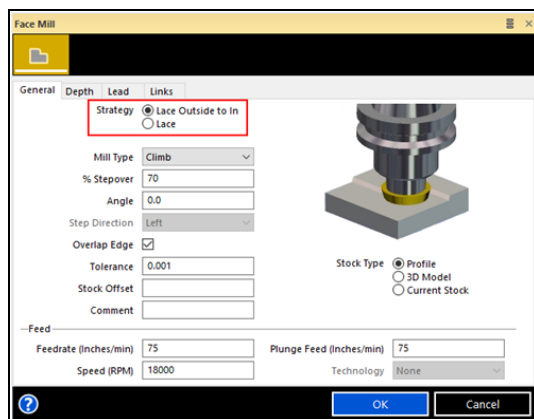
For this release, a **Strategy** option has been added to the General tab in the Face Mill cycle enabling you to choose between:

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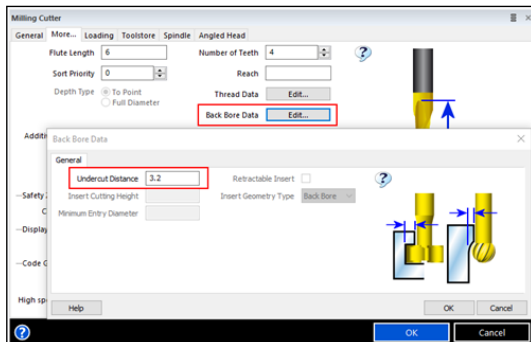
For this release, we have added support for Dovetail tools in the Profiling cycle:

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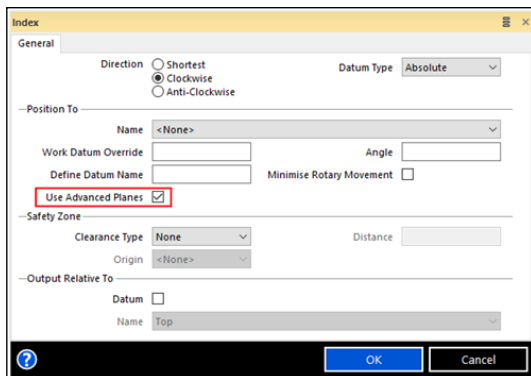
When indexing to a Workplane that would normally require it to be aligned to the machine axes, EDGECAM will leave it in its current orientation and moves and cycles will be relative to that plane.

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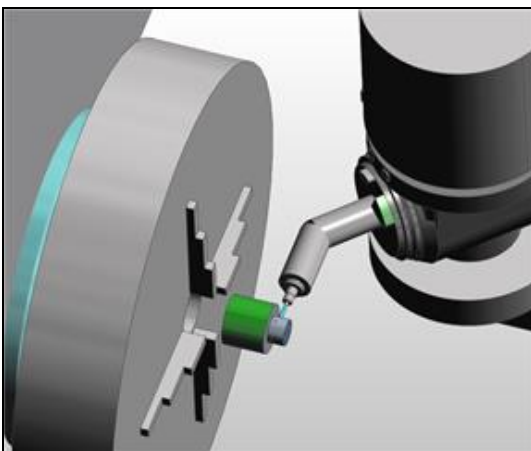
Previously, Angled Heads have only been available for Milling templates. This has now been extended for MTM based templates.

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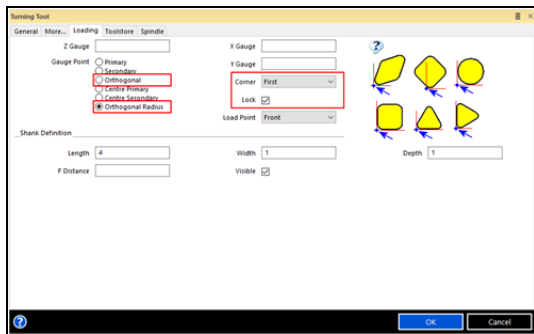
When rotating the insert (and shank), the system will automatically detect the gauge point but, occasionally, the user may want a different corner to be used. Therefore, to control the gauge point, additional controls have been added for this release:

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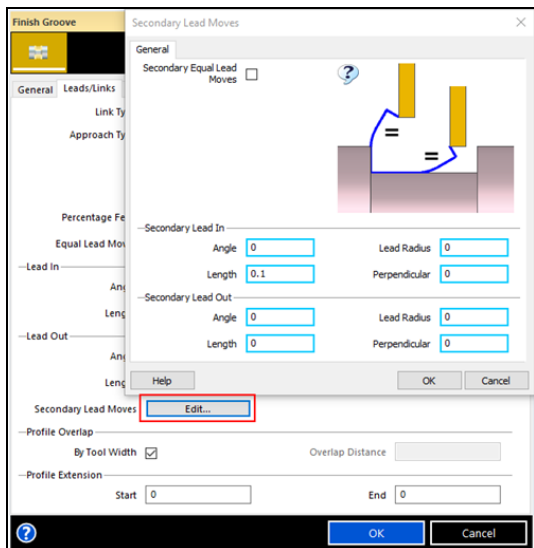
Secondary Lead Moves are now available on the Lead/Links tab of the Finish Groove cycle.

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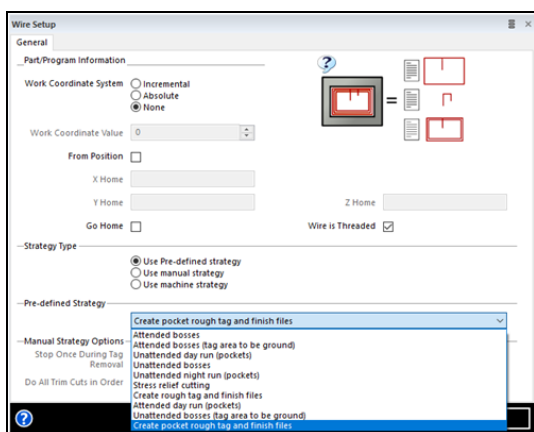
A **Create pocket rough tag and finish files** pre-defined strategy has been added to the **General** tab of the Wire Setup dialog.

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.

Performance improvements

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.
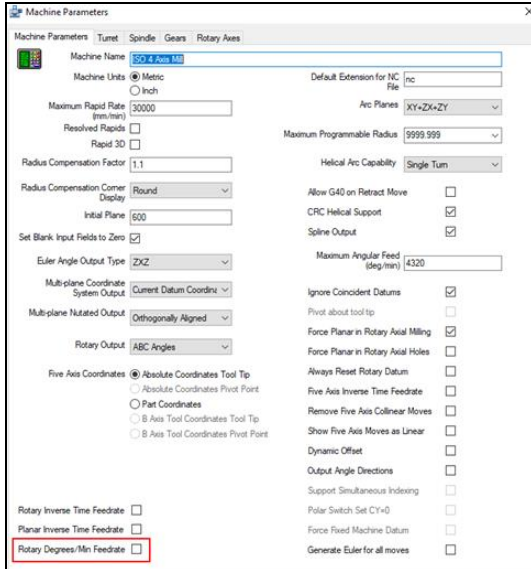
Preferences - Option to compress STL models in Simulator
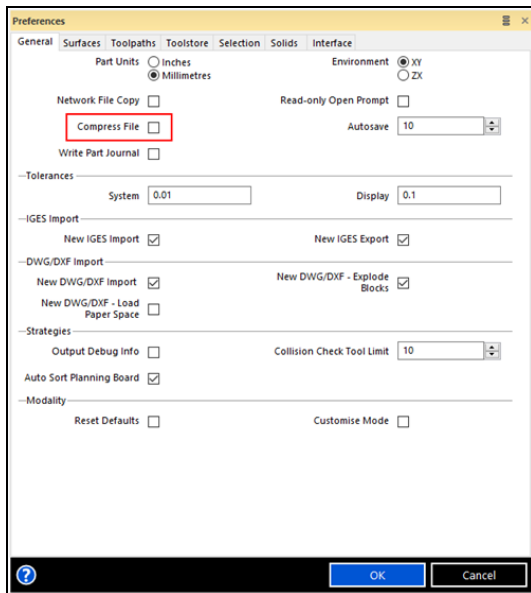
Simulator now stores the STL models generated by the stock and part solids during the simulation within the PPF file.

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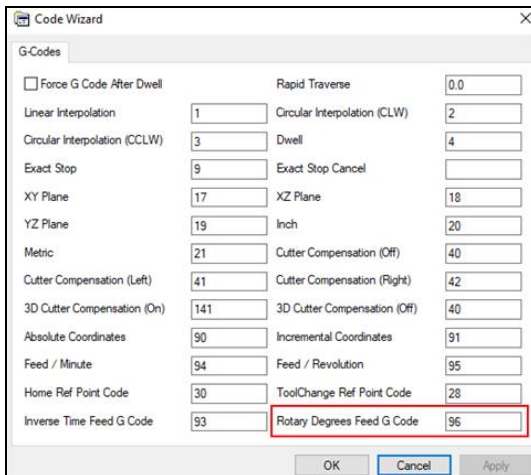
A **Rotary Degrees/Min Feedrate** option has been added to the **Machine Parameters** tab of the **Machine Parameters** dialog.

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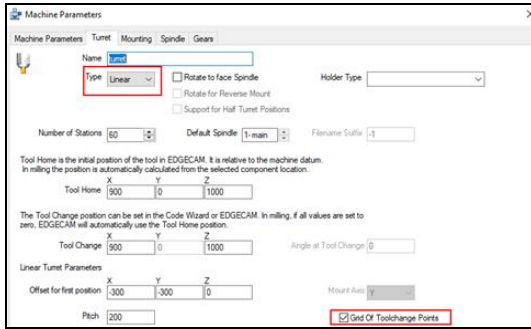
It is possible to change the value of the **Rotary Degrees Feed G Code** on the **G-Codes** tab of the **NC Style, G-Codes and Modality** section.

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.

Maintenance Database Report

For a full list of maintenance items resolved in EDGECAM 2021.0, please refer to the **Maintenance Database Report**.

New Features in Version 2020.1

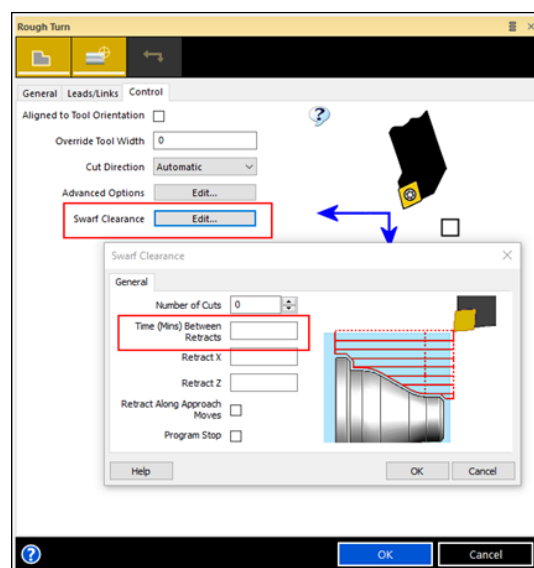Below is an overview of new features and enhancements in the last release.

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

Manufacture Enhancements

Rough Turn - Ability to define Swarf Clearance using elapsed cutting time For this release, a **Time (Mins) Between Retracts** option has been added to the Swarf Clearance dialog on the Control tab of the Rough Turn cycle. This option enables users to specify an elapsed cutting time, in minutes, to determine when to do the safe retract, as an alternative to the number of cuts. **Note:** Only actual contact toolpath elements contribute to the total cutting time and not LEAD in/out moves, rapid or feed links.	
Chamfering - Corner Strategy options added The **Corner Strategy** options from the Profiling cycle are now available in the Chamfering cycle enabling external sharp corners to be fully machined. The functionality should be the same as that available in Profiling. The **Corner Strategy** options have been added to the new Control tab in the Chamfering cycle.	
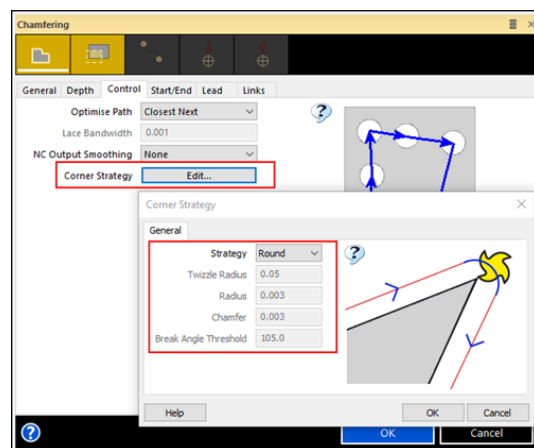
Chamfering - Optimise Path and Lace Bandwidth options added

The **Optimise Path** and **Lace Bandwidth** options from the Profiling cycle are now available in the Chamfering cycle. The functionality should be the same as that available in Profiling:
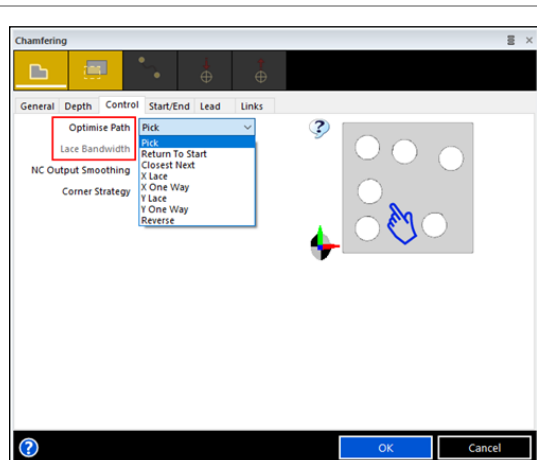
- The **Optimise Path** and **Lace Bandwidth** options have been added to a new Control tab in the Chamfering cycle.
- The **NC Output Smoothing** options have been moved from the General tab to the Control tab.

Note: Existing parts are set to use the **Pick Optimise Path** option.

Chamfering - Cut Increment and Z Offset options added

For this release, two new modifiers have been added to the Chamfering cycle in order to improve usability. Cut Increment has been added to the Depth tab and Z Offset has been added to General tab:

- The **Cut Increment** will work in conjunction with **Chamfer Depth**, starting from the top of the chamfer and repeating downwards until the set depth is reached.
- **Cut Increment** can be used with both wireframe and solid features. When using feature information, depth will be taken from that.
- The **Z Offset** allows the user to specify an offset on the part and is greyed out when **Chamfer Type** is set to **Deburring**.
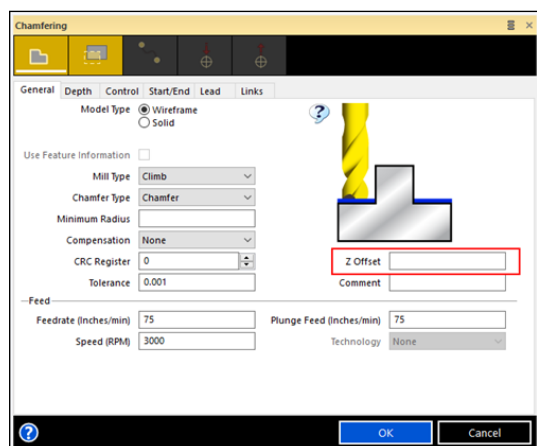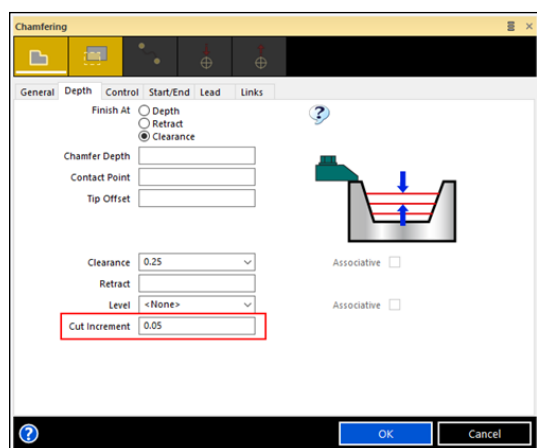
Chamfering - Control tab added and modifiers moved

In order to improve usage of the Chamfering cycle and avoid overloading the General tab, a Control tab has been added to which some modifiers have been moved.

Depth-related parameters (**Depth**, **Contact Point** and **Tip Offset**) have also been moved from the General tab to the Depth tab.
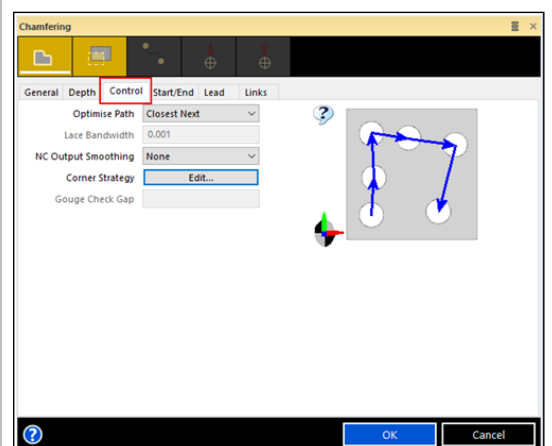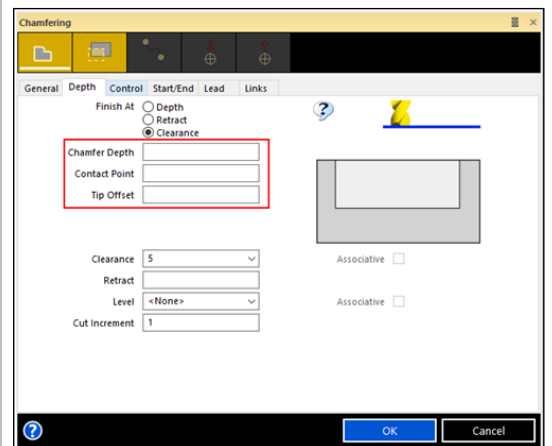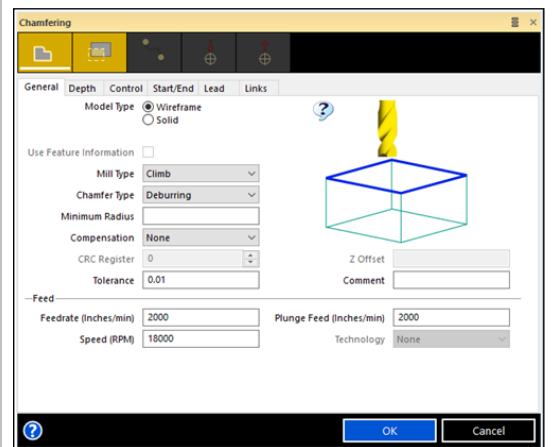
Parallel Lace - Extension options added

The ability to extend the toolpath has been added to the Parallel Lace cycle.

Four extension values have been added:

- **Start**
- **End**
- **Start Perpendicular**
- **End Perpendicular**

Start and **End** extensions are applied in the primary direction.

Start Perpendicular and **End Perpendicular** are only available and applied when **Perpendicular Lace** is enabled.

The extensions follow the direction of the cut (Climb/Conventional/Optimised).

The **Extensions** options have been added to the Control tab in the Parallel Lace cycle.
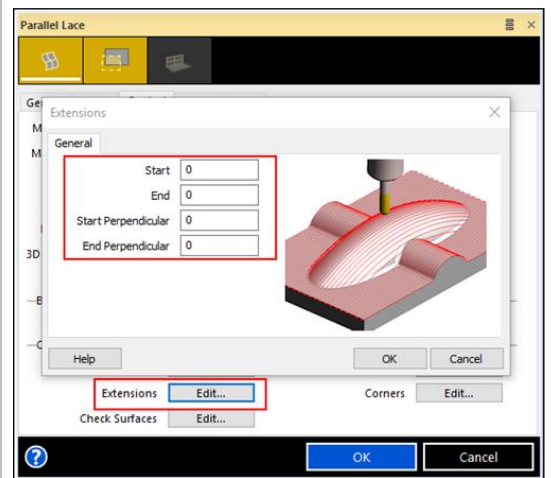
Additive Lace - Boundary control added

The ability to specify a boundary has been added to the Additive Lace cycle.

The **Tool Control** options are the same as for other cycles:

- **Inside**
- **Centre**
- **Outside**
- **Offset**

The **Boundary Control** options have been added to the General tab in the Additive Lace cycle.

Additive Lace - Maximum Overhang Angle implemented

When building up an element using additive, some negative angles can be applied to create undercut areas with overhangs. Different materials and processes result in varied negative build-ups. Therefore, when creating a wall that has a negative draft angle, it is now possible to specify the maximum angle overhang:
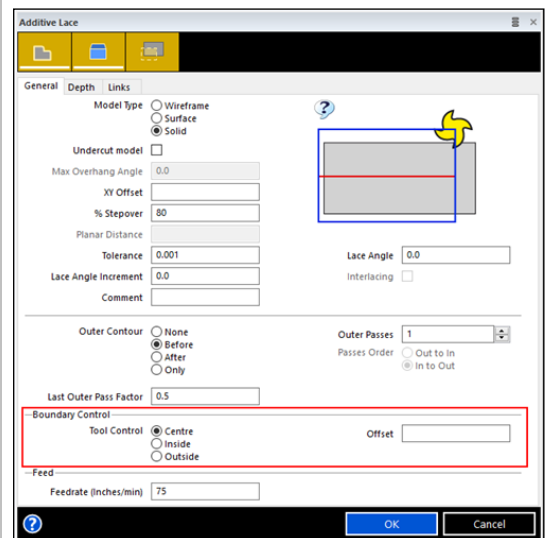
- If the target exceeds that angle, the cycle will add material to compensate.
- When a negative draft angle is encountered on all sides, the cycle will not currently allow full control and can, eventually, add more material than optimally expected.

The **Max Overhang Angle** option has been added to the General tab in the Additive Lace cycle and is only available when **Undercut Model** is selected.
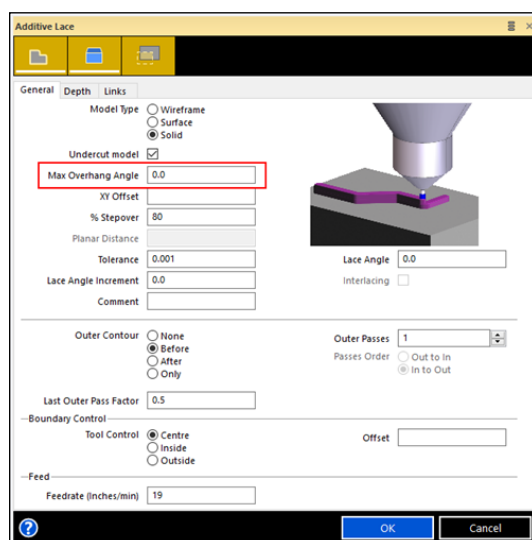
Slot Milling- Relax Gouge Check option added

The ability to relax gouge checking has been added to the Slot Milling cycle.

A new **Relax Gouge Check** option has been added to the General tab in the Slot Milling cycle.
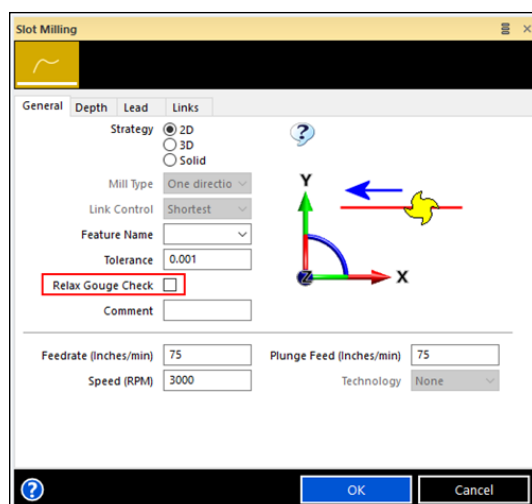
Face Mill - Horizontal lead type added

For this release, Lead **Type** options have been added to the Lead tab in the Face Mill cycle enabling you to choose between **Horizontal** and **Vertical** leads:

- Angle, radius and length are available for both lead types.
- Having 'rolling' leads can help increase the tool life.

Face Mill - Stock Type options added

For this release, the ability to use different stock types has been added to the General tab of the Face Mill cycle. The options work in the same way as for the Roughing cycle:

- **Profile** will allow the selection of a 2D profile and works in the same way as previous versions.
- **3D Model** allows the user to select the actual model
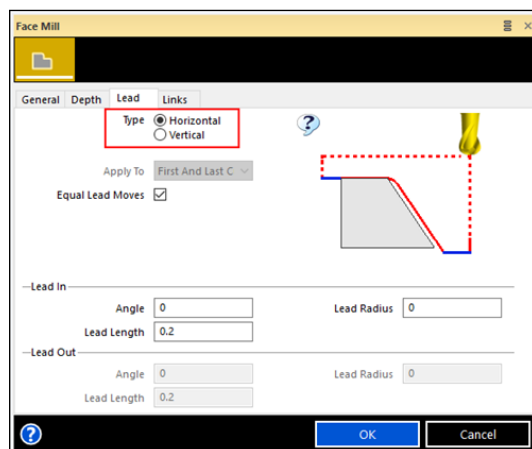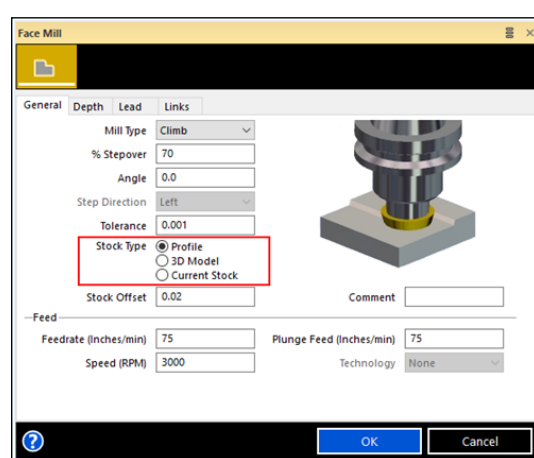- **Current Stock** will analyse the updated stock.

Create Sequence Fixture Tolerance

When the user creates a new sequence, stock is passed to the Simulator with a given tolerance. A tighter tolerance can be used for better representation and a relaxed tolerance can be used for better performance.

For this release, the same ability has been introduced for fixtures. The user can set a tolerance when creating the sequence and it will be used by the whole sequence:

- The **Tolerance** field has been renamed to **Stock Tolerance**.
- A new **Fixture Tolerance** has been added which defaults to 0.25 mm or 0.01 in.

Since most fixtures are prismatic, we expect a very small performance gain with the 0.25 mm default tolerance. However, we expect parts with complex and big fixtures to have improved performance when:

- Opening Simulator on first use.
- Playing Simulator.
- Using machining cycles that take the fixture in account when generating toolpaths.

Note: Only available when creating a New Milling Sequence.
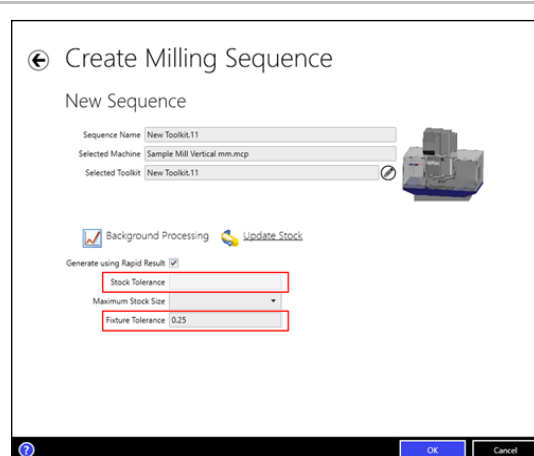
Solid Machinist for Creo and Granite changes

To help existing customers when changing from **Solid Machinist for Granite** to **Solid Machinist for Creo**, we have made the following changes:

Previously:

- A Creo solid loaded and saved in **Solid Machinist for Granite** (Granite body) could NOT be opened in **Solid Machinist for Creo**.
- A Creo solid loaded and saved in a **Solid Machinist for Creo** (Parasolid body) could NOT be opened in **Solid Machinist for Granite**.

Now:

- A Creo solid loaded and saved in **Solid Machinist for Granite** (Granite body) can NOT be opened in **Solid Machinist for Creo**. No change.
- A Creo solid loaded and saved in a **Solid Machinist for Creo** (Parasolid body) CAN be opened in **Solid Machinist for Granite**, modified and re-saved.

A new **Store Granite Solid As Parasolid** option has been added to the Solids tab of the Preferences dialog for **Solid Machinist for Granite**. This means the **Solid Machinist for Granite** and the **Solid Machinist for Creo** licenses can both open this PPF file and the solid is stored as a Parasolid body.

This change is only relevant to existing customers who have **Solid Machinist for Granite** licenses and have recently purchased one or more **Solid Machinist for Creo** licenses. This will allow a PPF file created in **Solid Machinist for Creo** to be loaded and edited in **Solid Machinist for Granite** (as a parasolid).
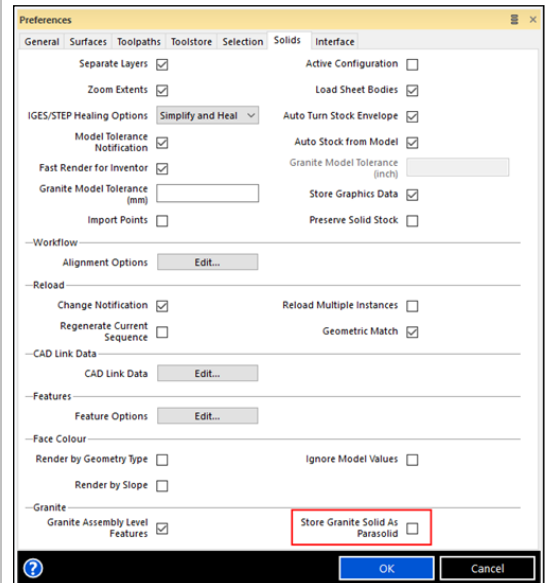
Finish Groove - Use Current Stock for Pre-Finish Plunge

Use Current Stock is now available when working with **Pre-Finish Plunge** on the General tab of the Finish Groove cycle.

Previously, the Clearance was based on the Feature height which, in some cases, is not always desirable:

- When the Stock is higher than the Feature, for example, the result would be a tool collision with the Stock, depending on the Clearance value. This can now be avoided with **Use Current Stock**.
- The new option is also useful when the groove has already been machined by a previous cycle.
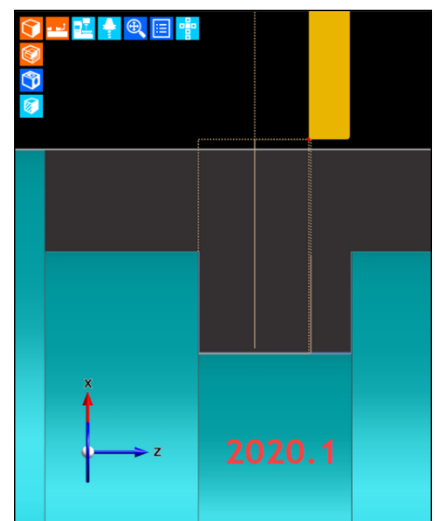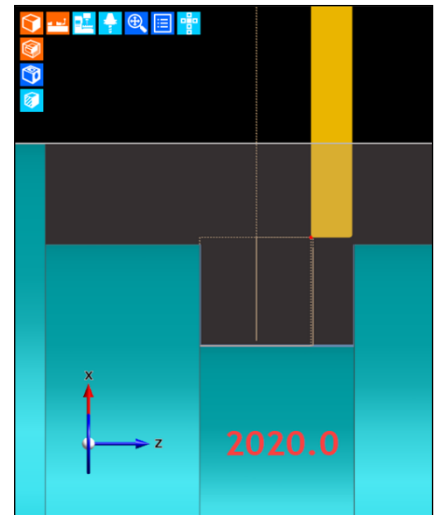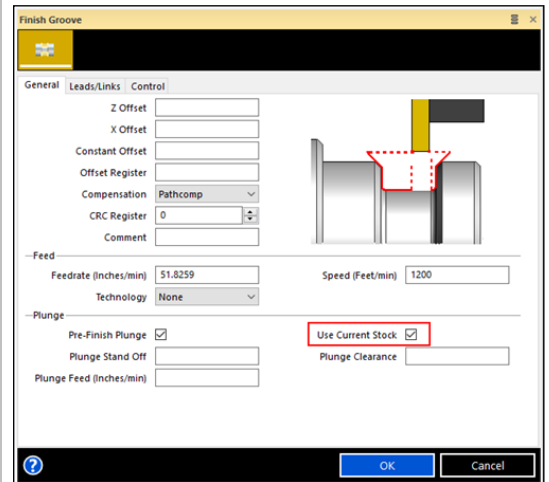
Prevent Switching Sequences when Background Processing

It is known that EDGECAM only supports a single session (one instance of the software open) because of conflicts in the Cache folder. It has been identified that a single session with multiple sequences could show similar results if the user has **Set Safe Start** point turned on. The user can regenerate a whole sequence, switch to another sequence and regenerate it as well, leading to conflicts; especially with Index and similar cycles or if the user decides to delete the cache while another sequence is still regenerating.

To prevent these issues from happening, we have disabled the ability to switch sequences while background processing is active (machining cycles or Update Stock calculating) and the following message will be displayed:

```
Cannot leave current sequence while background processing is in progress
```

Note: This limitation should be removed once we support multiple machining sequences regenerating toolpaths.

EDGECAM Inspect improvements

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

- **New Inspect feature engine**

  The EDGECAM Inspect feature engine has been redeveloped. Inspect features are now treated in a similar manner to normal EDGECAM features. When a feature is created, it is shown on the Features browser and, by double-clicking, you can edit any feature and change parameters.

  Note: Due to the Inspect features redevelopment, features created in old EDGECAM versions will need to be recreated.

- **Gap feature**

  The **Gap** feature allows the user to create a feature to measure distance between two parallel walls by defining two points.

- **Normalize option**

  If the **Normalize** option in the Results dialog is selected, the programmed value, deviation, and tolerances are calculated from the centre of the tolerance zone.

  For example, by default (option unchecked), if the programmed X position is 100, measured is 100.01 and tolerances are -0.02/0, then the deviation is +0.01. However, if the option is checked, then the programmed value considered for the deviation calculation is 99.99 (the middle value between 99.98 and 100) and the deviation is +0.02.

- **EDGECAM Inspect Probing Cycle and post processors**

  Due to feedback from the previous release, the EDGECAM Inspect Probing Cycle has been redeveloped with new functionality which allows a wider range of applications.

  The reorganisation of the Probing Cycle has required changes to the Code Generator system variables which means that existing post processors with Inspection functionality must be updated to the latest version. Otherwise, the old post with the unchanged system variables will fail to compile.

  If you are already an Inspect Probing Cycle (Inspect Canned Cycle) user, you will need to review all the Probing Cycle information from your post. In addition, old post processors that used Inspection will, at the update, require .cgs customisation to correct some format table entries.

  The cycle re-design will provide a solid basis for future work with Inspection and this should be the only time that an update is necessary.

- **Inside and Outside features – Find closest parallel wall**

  The Inside and Outside features have been improved and now these two feature types can be created even when there are obstacles between the parallel walls, for example, a boss inside a pocket or a hole between two walls.

- **Inspection Cycle – Calibration datum name**

  A **Datum Name** option has been added to the Advanced tab of the Inspection Cycle. This option allows the user to define or select a datum at which the sphere is positioned.

  If blank, the calibration datum name is compiled using the following logic:

  "clb + touch datum name + _ + calibration sphere position".

  For example, "clbComponentDatum_(-200/0/0)".

  If not blank, the calibration datum name uses the following logic:

  - If datum for touches is the initial datum, Datum Name is used as is.
  - Otherwise, the calibration datum name is the set Datum Name appended with the touch datum name.

- **Filters added to the Clear All command**

  The **Clear All** command has been improved and the user can now filter which feature type is to be removed.

Tombstone Enhancements

Tombstone Ordering

Previously, the user could select the Ordering for the Tombstone tools to be either:

- **By Tool** - Rationalises by tool and then reduces Indexes.
- **By Plane** - Rationalises by Plane/Datum and then reduces toolchanges.

For this release, two new options have been added and the two old ones renamed to provide greater clarity for the user:

- **By Tool then by Plane**
- **By Plane then by tool**
- **By Tool Only - Keep sequence order**
- **By Plane Only - Keep sequence order**

The new options should allow the user to have more control over the machining sequence and to avoid collisions.

Tombstone					✕
Add Tombstone	Add Part	Copy	Delete	Edit	Reload
Inserted Parts		Sequence	Mate	Target	Datum
Configuration					

Fixture Clearance Options | Ordering | Output Options

○ None
● By Tool then by Plane
○ By Plane then by Tool
○ By Tool Only - Keep sequence order
○ By Plane Only - Keep sequence order

Apply | Cancel
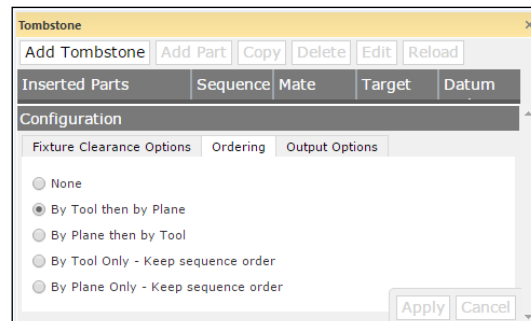
Second Sequence

Some Tombstone parts have multiple sequences which caused issues previously when adding them to the Seed file. This is properly supported in this release:

- The active sequence when the part is saved will generate the graphics for the Windows Explorer Preview which means that only a single sequence will be displayed. This prevents multiple fixtures from being displayed and the toolpath being incorrect.
- Once the part is added to the Tombstone Browser, only the graphics related to the sequence selected will be displayed.
- Issues with fixtures not being rendered at the proper location have been resolved for this release.

Simulator Enhancements

Mouse Controls in Simulator

Simulator has been updated to follow the **Use Designer Mouse Controls** option in the Preferences dialog of EDGECAM.

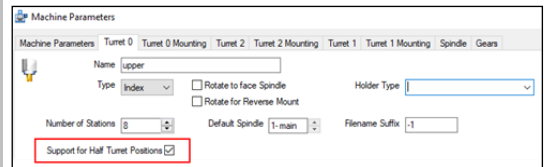With this preference on, the Pan and Zoom controls change in the same way as EDGECAM.
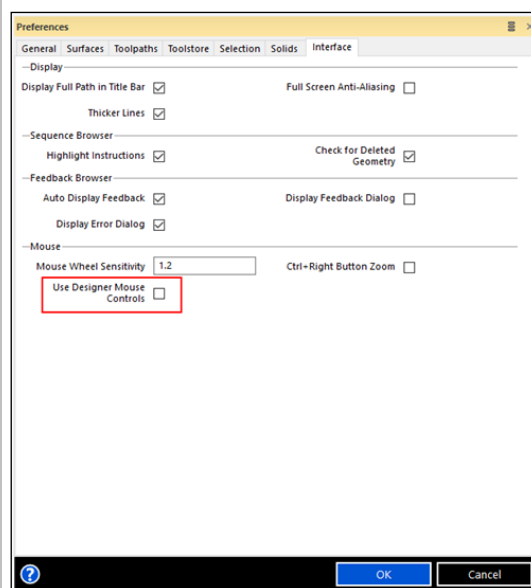
Code Wizard Enhancements

Turning - Ability to use turrets with half positions

Users are now able to set turrets to support Half Turret Positions in the Code Wizard and EDGECAM.

A **Support for Half Turret Positions** option has been added to the **Turret** tab of the **Machine Parameters** dialog.

The option offers a standard solution for the Half Turret Positions when compared with the regular positions of the turret, which is referred to as the **Code Wizard Position**. This solution can be changed by the user according to their requirements. In the EDGECAM Simulator, the Half Turret Positions will be displayed as set on the **Real Machine Position**.

Post Processor / Dwell for Through Tool Coolant activation

For this release, a new **Through Coolant Dwell** setting has been added to all post processor templates:

- When it is not zero, this will be the dwelling time when activating the Through Tool Coolant.
- A post action has been configured in the COOLTHROTOOL token (used for the M code of Through Tool Coolant), generating the output of the respective dwell value immediately after it has been activated.