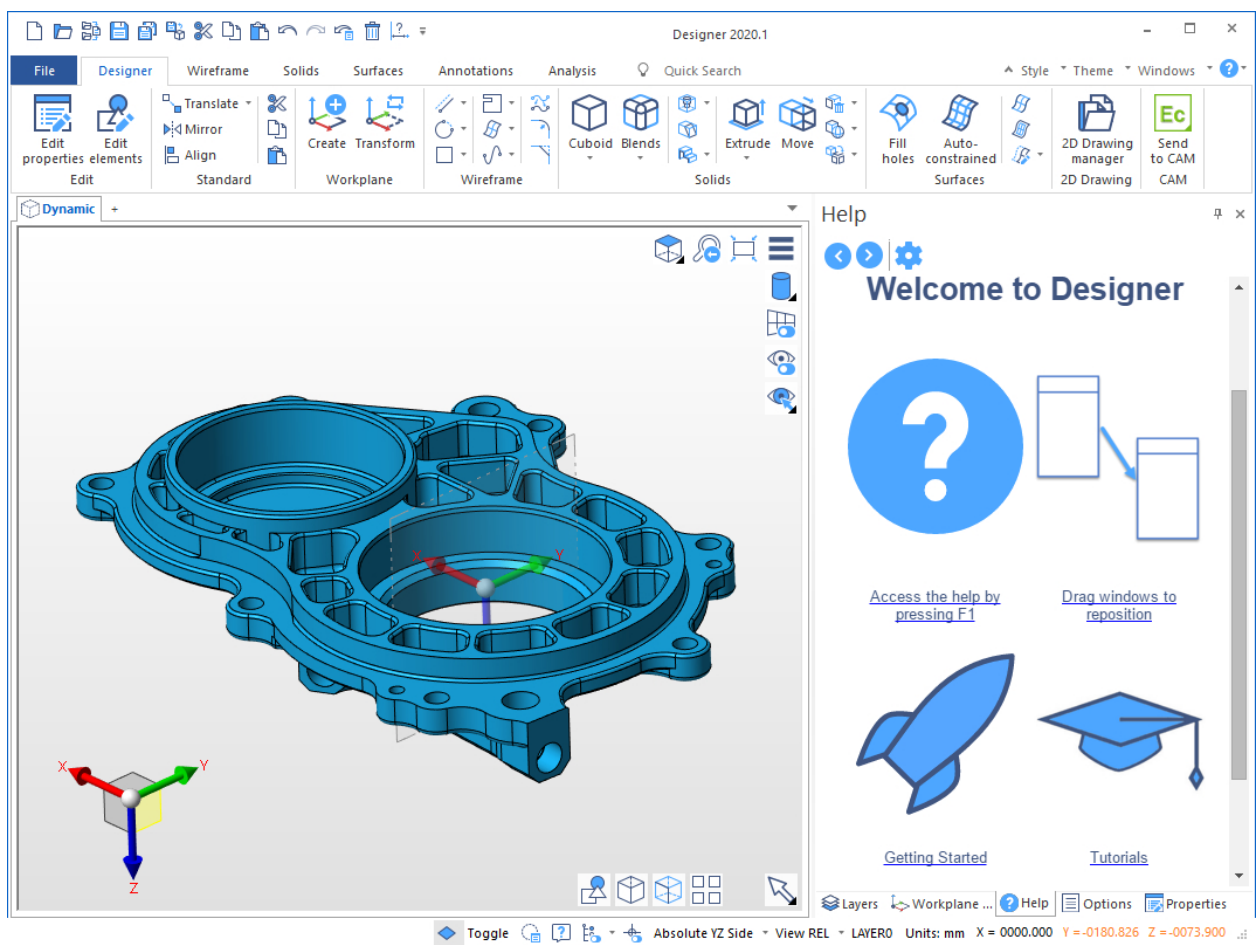


What's New in EDGECAM 2020.1



This document highlights new product features and enhancements in EDGECAM 2020.1.

To run EDGECAM 2020.1, the maintenance expiry date in the license must be October 2019 or later.

Contents

'WHAT'S NEW' DOCUMENT OVERVIEW	3
IMPORTANT INFORMATION	4

Manufacture Enhancements

ROUGH TURN - ABILITY TO DEFINE SWarf CLEARANCE USING ELAPSED CUTTING TIME	5
CHAMFERING - CORNER STRATEGY OPTIONS ADDED	5
CHAMFERING - OPTIMISE PATH AND LACE BANDWIDTH OPTIONS ADDED	6
CHAMFERING - CUT INCREMENT AND Z OFFSET OPTIONS ADDED	6
CHAMFERING - CONTROL TAB ADDED AND MODIFIERS MOVED	7
PARALLEL LACE - EXTENSION OPTIONS ADDED	8
ADDITIVE LACE - BOUNDARY CONTROL ADDED	8
ADDITIVE LACE - MAXIMUM OVERHANG ANGLE IMPLEMENTED	9
SLOT MILLING- RELAX GOUGE CHECK OPTION ADDED	9
FACE MILL - HORIZONTAL LEAD TYPE ADDED	10
FACE MILL - STOCK TYPE OPTIONS ADDED	10
CREATE SEQUENCE FIXTURE TOLERANCE	11
SOLID MACHINIST FOR CREO AND GRANITE CHANGES	12
FINISH GROOVE - USE CURRENT STOCK FOR PRE-FINISH PLUNGE	13
PREVENT SWITCHING SEQUENCES WHEN BACKGROUND PROCESSING	14
EDGE CAM INSPECT IMPROVEMENTS	14

Tombstone Enhancements

TOMBSTONE ORDERING	16
SECOND SEQUENCE	16

Simulator Enhancements

MOUSE CONTROLS IN SIMULATOR	17
-----------------------------------	----

Code Wizard Enhancements

TURNING - ABILITY TO USE TURRETS WITH HALF POSITIONS	18
POST PROCESSOR / DWELL FOR THROUGH TOOL COOLANT ACTIVATION	19
MAINTENANCE DATABASE REPORT	20
NEW FEATURES IN VERSION 2020.0	21

'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 2020.0](#).

For a summary of new features in previous releases, please visit the [History section of the EDGECAM website](#).

Important Information

Part Modeler has been retired

For this release, Part Modeler will not be shipped with the product. The last installed Part Modeler will launch EDGECAM 2020.1. We recommend that our customers move on to Designer.

Windows 7 and 8.1 Support

Microsoft will be ending extended support of Windows 7 by January 2020.

Windows 8.1 mainstream support ended in 2018 and is used by less than 5% of our users.

EDGECAM 2021.0 (April 2020) will not support Windows 7 or 8.1. It will install, but the user will be warned that those operating systems are not supported.

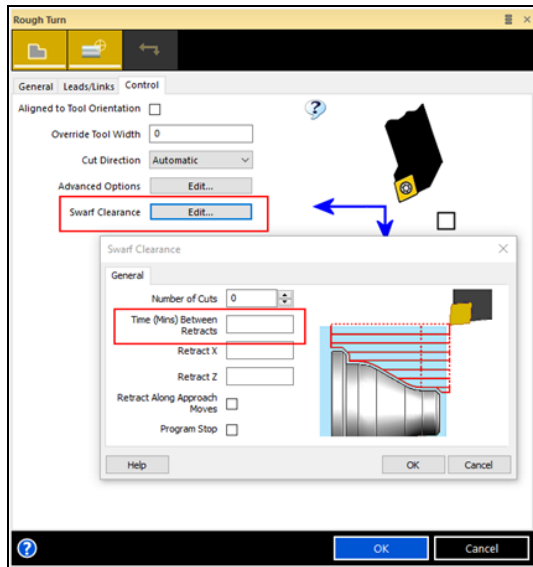
We would recommend updating to Windows 10 Build 1803 or later.

EDGECAM Inspect post processors

Inspection functionality will need to be updated to the latest template version. Otherwise, the old post with the unchanged system variables will fail to compile.

Manufacture Enhancements

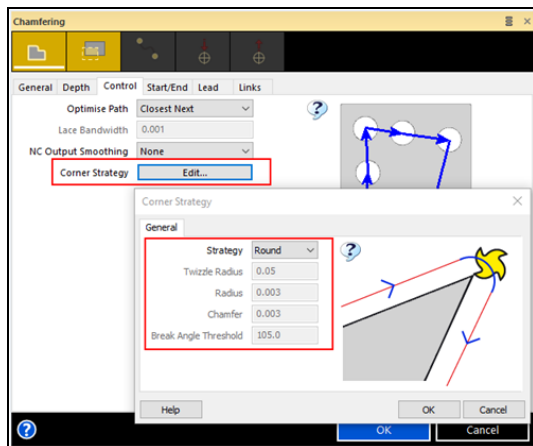
Rough Turn - Ability to define Swarf Clearance using elapsed cutting time



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.

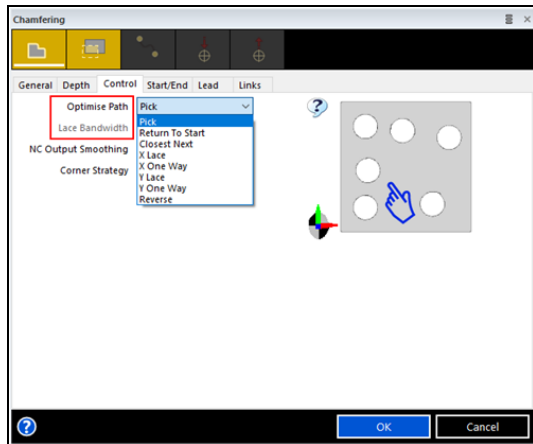
Chamfering - Corner Strategy options added



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

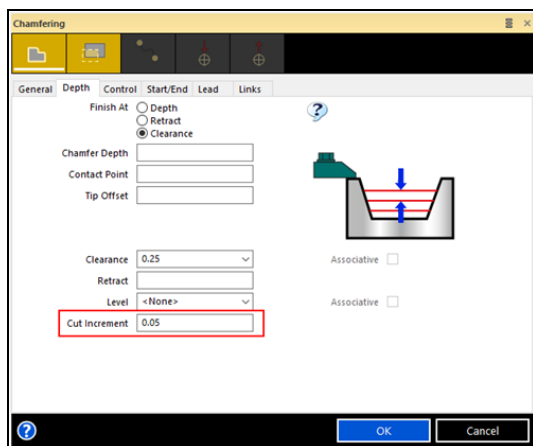


The **Optimise Path** and **Lace Bandwidth** options from the Profiling cycle are now available in the Chamfering cycle. The functionality should be the same as that available in Profiling:

- 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.

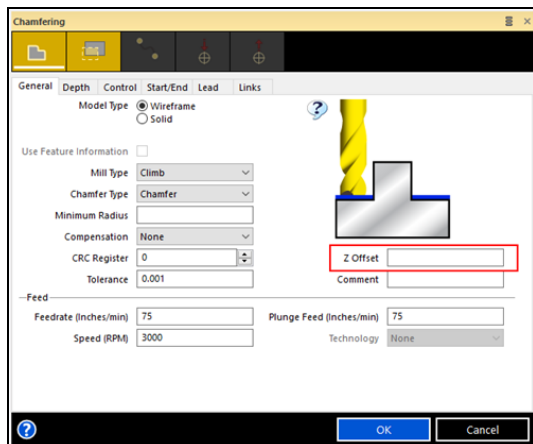
Note: Existing parts are set to use the **Pick Optimise Path** option.

Chamfering - Cut Increment and Z Offset options added

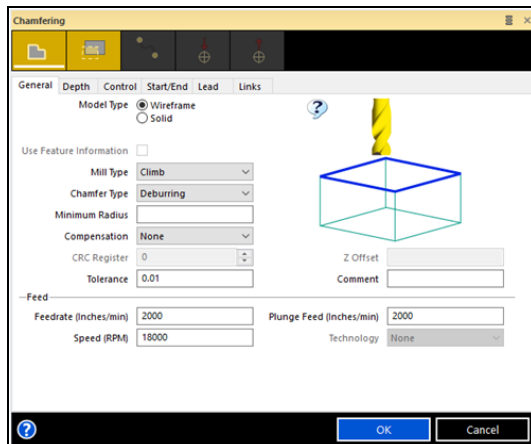


For this release, two new modifiers have been added to the Chamfering cycle in order to improve usability. **Cut Increment** has been added to the Depth tab and **Z Offset** has been added to General tab:

- 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**.

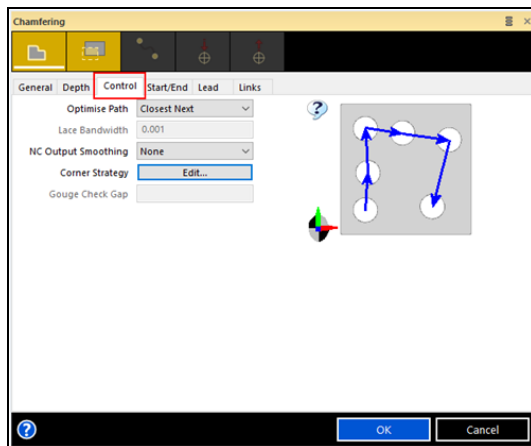
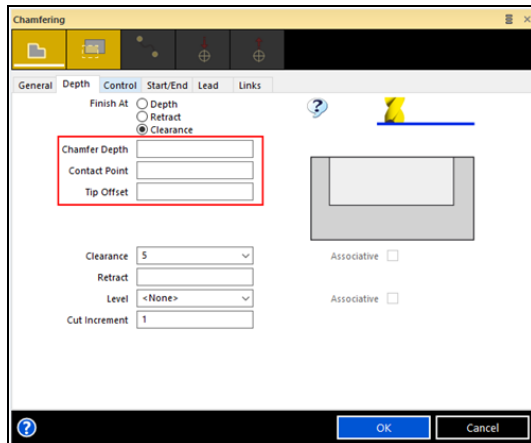


Chamfering - Control tab added and modifiers moved

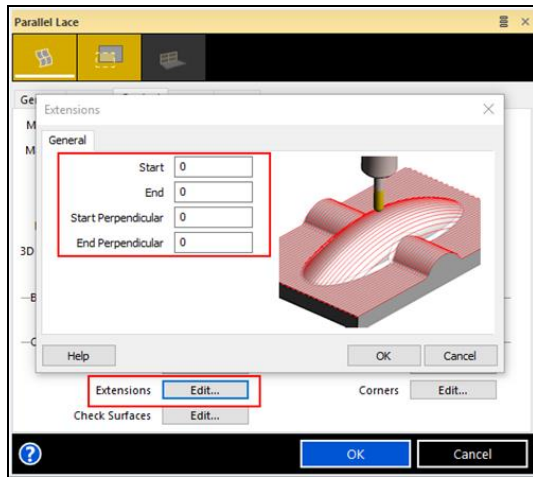


In order to improve usage of the Chamfering cycle and avoid overloading the General tab, a Control tab has been added to which some modifiers have been moved.

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**

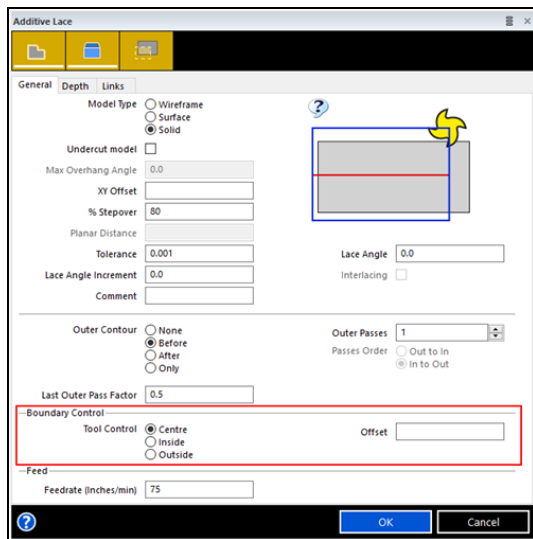
Start and **End** extensions are applied in the primary direction.

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



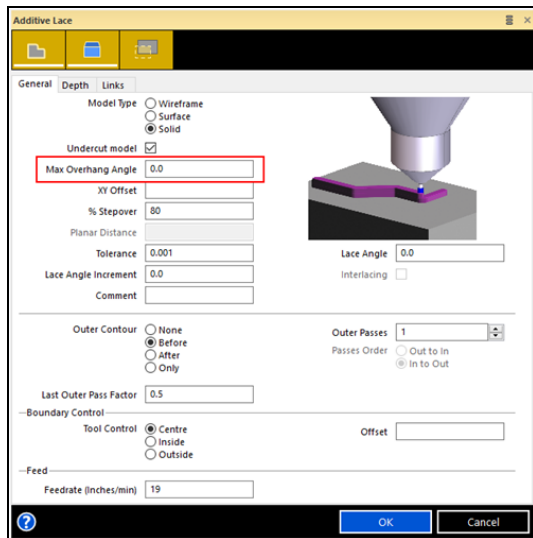
The ability to specify a boundary has been added to the Additive Lace cycle.

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

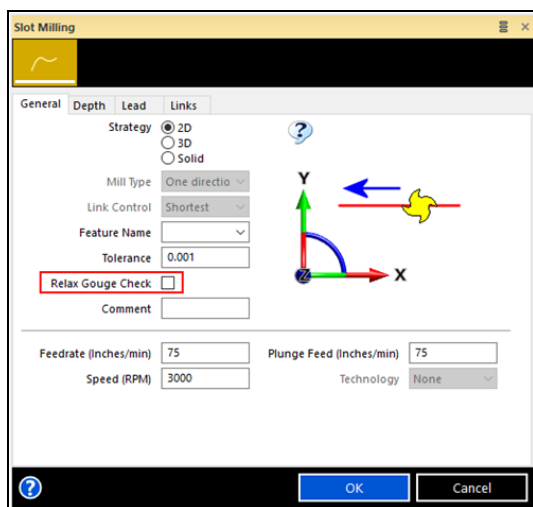


When building up an element using additive, some negative angles can be applied to create undercut areas with overhangs. Different materials and processes result in varied negative build-ups. Therefore, when creating a wall that has a negative draft angle, it is now possible to specify the maximum angle 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.

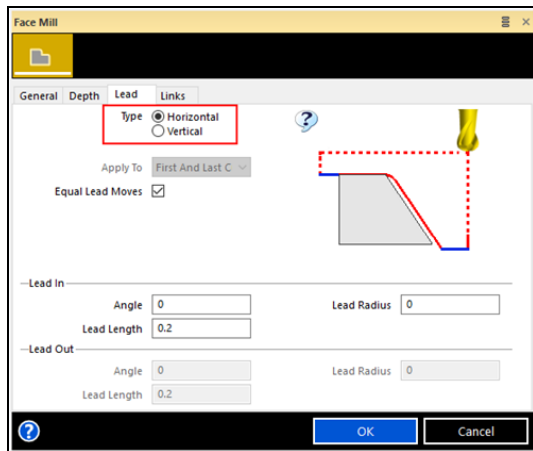
Slot Milling- Relax Gouge Check option added



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.

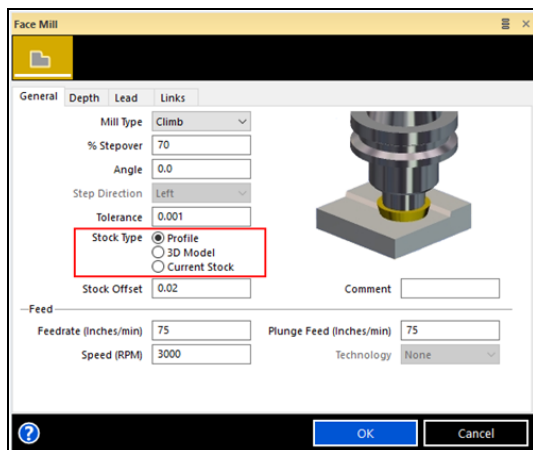
Face Mill - Horizontal lead type added



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.

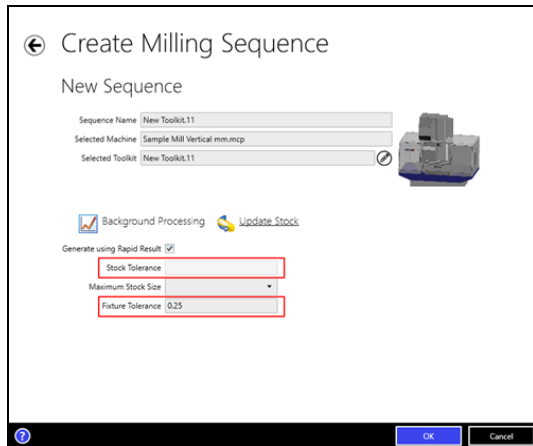
Face Mill - Stock Type options added



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:

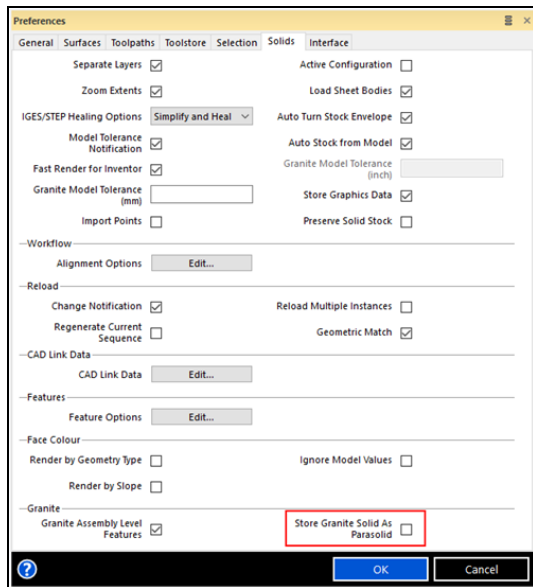
- The **Tolerance** field has been renamed to **Stock Tolerance**.
- A new **Fixture Tolerance** has been added which defaults to 0.25 mm or 0.01 in.

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**.

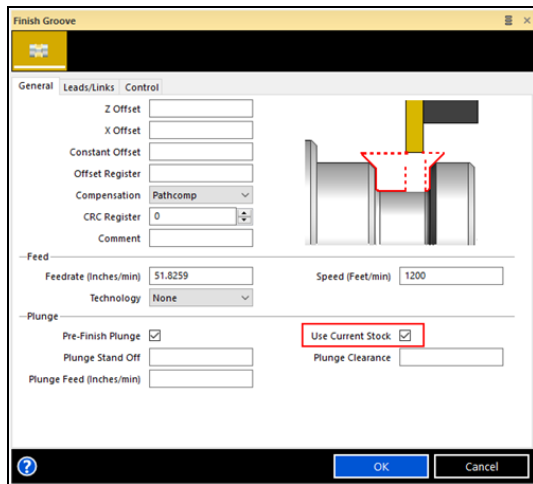
Now:

- 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).

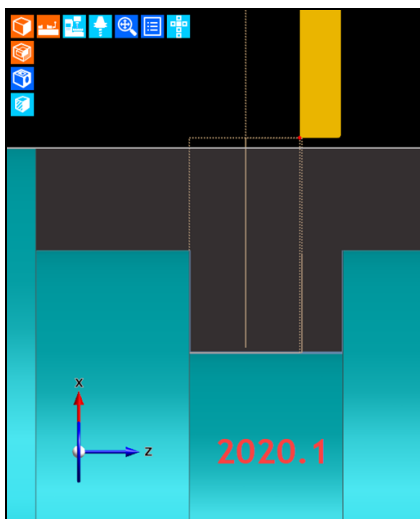
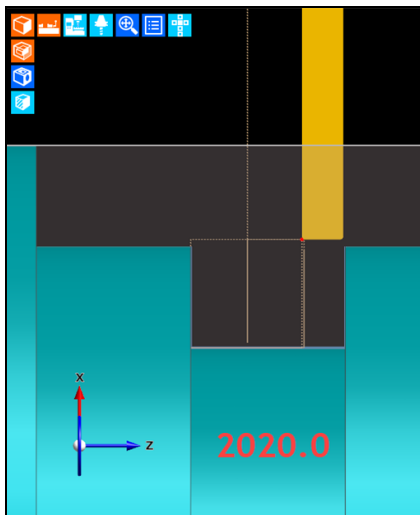
Finish Groove - Use Current Stock for Pre-Finish Plunge



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 EDGE CAM 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.

EDGE CAM Inspect improvements

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

- **New Inspect feature engine**

The EDGE CAM Inspect feature engine has been redeveloped. Inspect features are now treated in a similar manner to normal EDGE CAM 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 EDGE CAM 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.

- **EDGE CAM Inspect Probing Cycle and post processors**

Due to feedback from the previous release, the EDGE CAM 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:

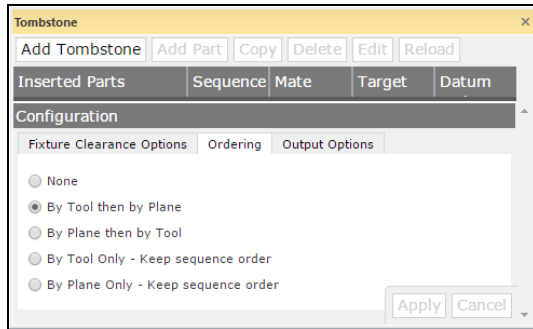
- If datum for touches is the initial datum, Datum Name is used as is.
- Otherwise, the calibration datum name is the set Datum Name appended with the touch datum name.

- **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.

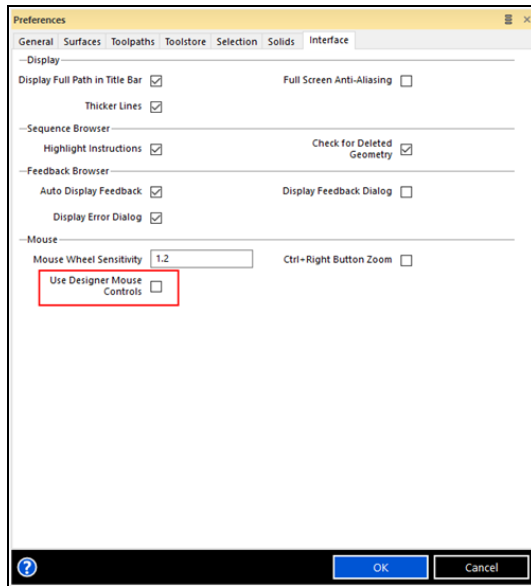
Second Sequence

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

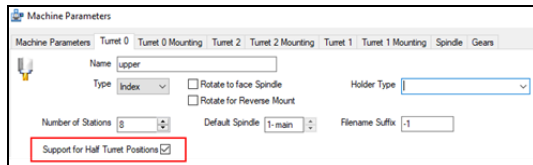


Simulator has been updated to follow the **Use Designer Mouse Controls** option in the Preferences dialog of EDGE CAM.

With this preference on, the Pan and Zoom controls change in the same way as EDGE CAM.

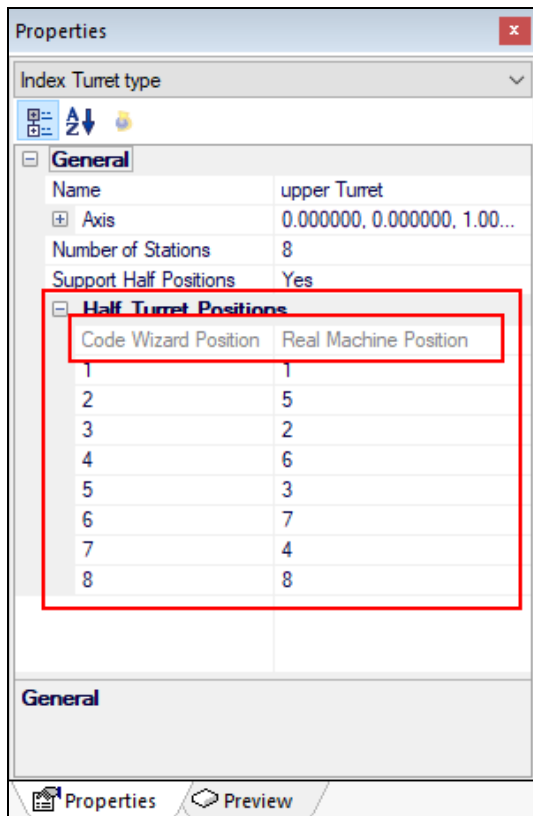
Code Wizard Enhancements

Turning - Ability to use turrets with half positions

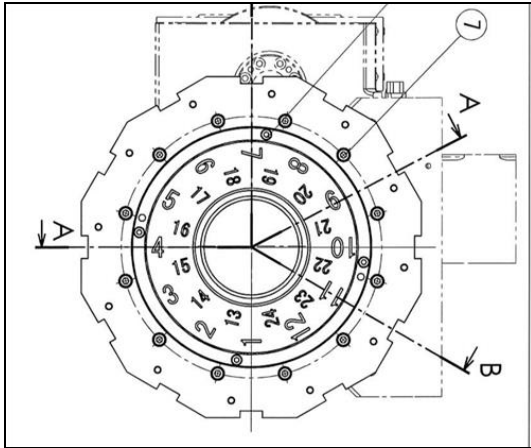


Users are now able to set turrets to support Half Turret Positions in the Code Wizard and EDGECAM.

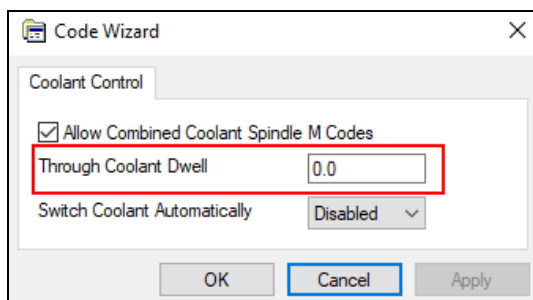
A **Support for Half Turret Positions** option has been added to the **Turret** tab of the **Machine Parameters** dialog.



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.

Maintenance Database Report

For a full list of maintenance items resolved in EDGECAM 2020.1, please refer to the [Maintenance Database Report](#).

New Features in Version 2020.0

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

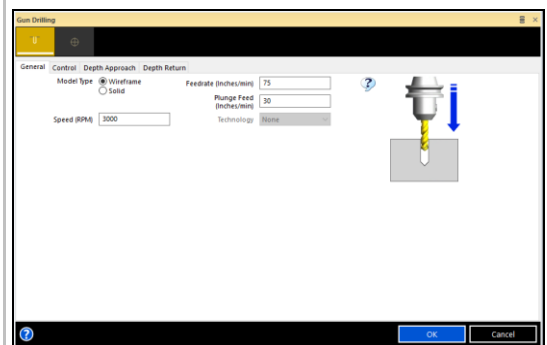
For a summary of new features in previous releases, please visit the [History section of the EDGECAM website](#).

Manufacture Enhancements

New Gun Drilling Cycle

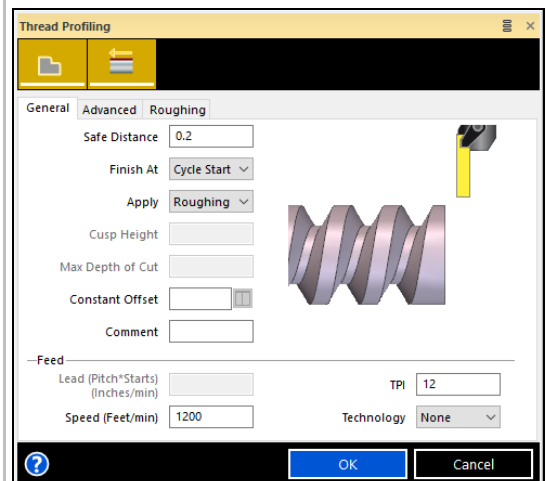
EDGE CAM now provides a Gun Drilling cycle. This dedicated interface aims to simplify the information contained in the dialog and provide the same functionality as in the Deep Hole tab of the Hole cycle but with some additional controls.

Note: The Deep Hole tab in the Hole Cycle dialog will become obsolete and will be retired in the near future.



New Thread Profiling Cycle

EDGE CAM now provides a Thread Profiling cycle. This new cycle has been implemented for this release providing a way to machine large screw threads when the profile is too large to cut with a regular threading insert.



General Quill Enhancements

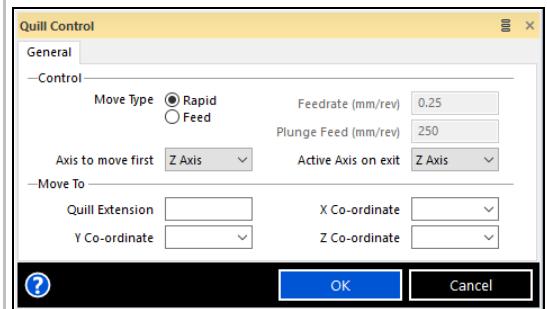
For this release, a new **Quill Control** dialog and the possibility to add auxiliary Z axes to MTM post processors have been introduced:

Quill Control

The new **Quill Control** dialog on the **Move Tool** menu provides centralised control over the Auxiliary axis moves, such as which axis is active after the move is completed and which axis (main or auxiliary) should move first.

Tracking Window

A new feature has been added to increase the control over the position of the quill, which is the possibility to track its current position through the Tracking window. This is useful when simulating the auxiliary axis movements internally. The same enhancement applies to auxiliary X axes.

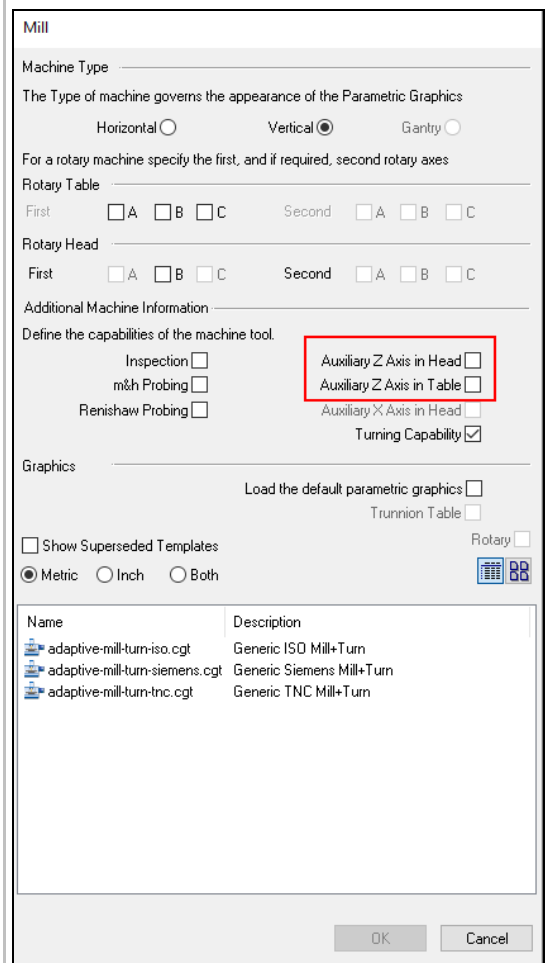


Quills on Multi Task Machines

Auxiliary Z axes

Auxiliary Z axes can now be added when building MTM post processors. Previously, the Auxiliary Axes options were unavailable when Turning Capability was selected.

It is possible to add these auxiliary devices to Vertical and Horizontal Mill Turn machines, either in the Table or in the Head. This allows for the support of VTL Crossrail configurations.



New Milling Tool Types

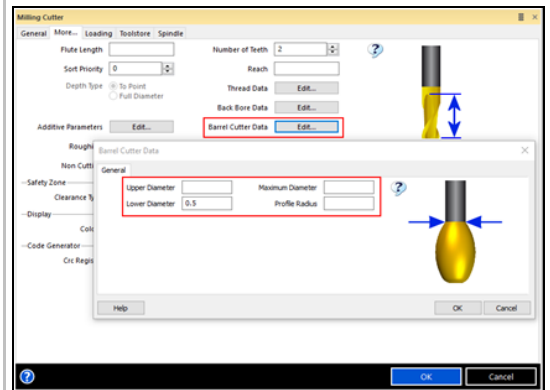
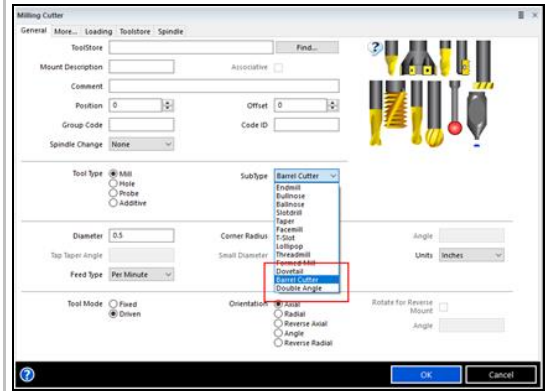
For this release, three Milling Tool Types have been added which can be selected from the General tab of the Milling Cutter dialog:

- Barrel Cutters for 5 Axis Machining.
- DoveTail and Double Angle Tools for Chamfer generation.

For Barrel tools, the Barrel Cutter Data dialog can be accessed from the More tab allowing the **Upper Diameter**, **Lower Diameter**, **Maximum Diameter** and **Profile Radius** to be specified.

For Dovetail and Double Angle tools, **Diameter** and **Angle** can be specified on the General tab; **Flute Length** on the More tab. Note that the **Included Angle** setting in the ToolStore is named **Angle** in EDGE CAM.

For all tools, you can, optionally, specify a **Corner Radius**.



Edgecam Inspect improvements

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

- **Updates on the Ribbon**

In order to follow the EDGECAM Workflow, the EDGECAM Inspect commands have been reorganised in the following areas of the ribbon:

- **Features tab.**
- **Machining tab.**
- **NC Code tab.**

- **Split Options menu**

The **Options** menu on the Features tab is now called **Inspection Defaults** and contains all of the default options for new features and tolerances.

A second Options menu (**Output Options**) has been added to the NC Code tab which contains all of the output options.

- **Settings related to report on the Results dialog**

The options related to reporting can now be controlled directly from the Results dialog.

- **Safe Retract controls**

The Inspection Cycle now provides options to control what the cycle does before indexing the probe tool and at the end of the cycle.

- **Ability to use the Alternate Solution**

EDGECAM Inspect now offers an option to control the way that the probe indexes to a feature.

This option adds another rotation to the index move by rotating the workplane and, therefore, the part, through 180°, the prefix 'alt' is added to this new workplane.

- **Export / Import points**

Two new commands have been added to EDGECAM Inspect; Export and Import points:

- **The Export command allows the user to export points to any text format (usually .xml file) and apply a processor (plugin) to translate them.**
- **The Import command allows the user to import points from other applications such as PCDMIS and 3DFI. The command reads the XML file created by a processor from a text file.**

- **Improvements in the Angle To Axis feature**

The **Angle To X Axis**, **Angle To Y Axis** and **Angle To Z Axis** commands have been replaced by a single **Angle To Line** command.

The **Reference Axis** is set automatically depending on the command picks but, if necessary, it can be changed by editing the feature. In addition, the feature direction can be controlled while creating the feature; the second pick determines the feature direction.

- **Changes to Constructed features**

- **Constructed features have been renamed.** Distance To Line is Distance and Relative Angle is Angle.
- **Line can be created for more than two single-pointed features allowing Straightness to be evaluated:**
Note: To evaluate Straightness, create the line feature by picking more than two single-pointed features and then check the Straightness option by editing the feature.
- **Distance can be applied to any combination of single-pointed, constructed line and plane. We can measure distance between points, distance to line, distance between lines, distance to plane and distance between planes.**

- **Convert Circular feature into Arc feature**

Circular features can be converted into Arc features, and Arc features can be converted into Circular features, by editing the **End Angle**:

- If End Angle is **blank**, the feature is treated as a **Circle**.
- If End Angle is **not blank**, the **Circle** is converted to **Arc**.

- **Allow index for non-indexable**

This improvement allows the user to re-assign Cone, Cylinder, Hole and Rectangle features to any other workplane if the plane of the workplane is parallel to the feature. It may be very useful, for example, to re-assign features in the sub spindle.

- **Option to use last block of data (between operators 14 and 9) in the result file**

EDGE CAM Inspect now uses the last block of data (block between operators 14 and 9) in the journal file to evaluate results.

It is an important improvement because some controllers do not have a function to delete the journal file and it may contain several blocks of results where the last one is the most important.

- **Associative to Feature when no solid**

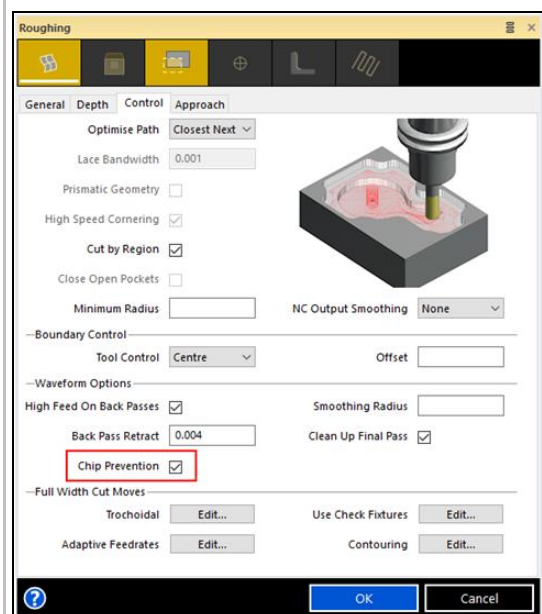
Associative clearance can now be used in the Inspection Cycle even without a solid. When **Associative** is On, the clearance level is calculated relative to the highest point of the feature.

Roughing Cycle - Rough Waveform Chip Prevention option added

When roughing boss regions, the toolpath can generate collapsed regions that form a peg of material. These thin walls become weak, vibrate and break off, potentially causing tool damage.

For this release, we have introduced a **Chip Prevention** option to the Rough Waveform strategy.

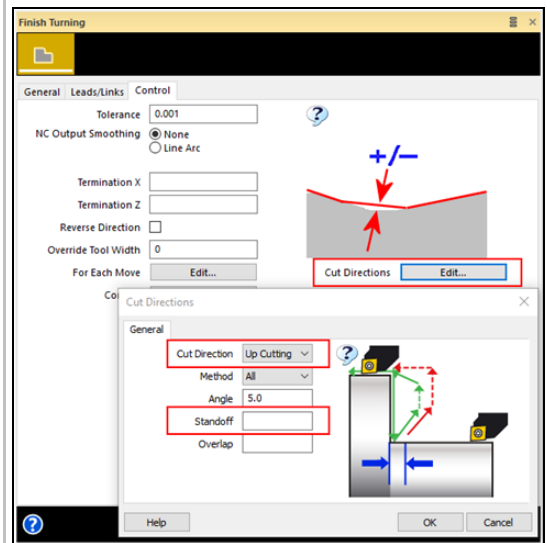
Selecting Chip Prevention enables Across moves to remove narrow bits of stock and prevent them breaking off.



Finish Turning - Stand Off available for Up Cutting

The **Stand Off** option is now available for **Up Cutting** or **Down Cutting**. Previously, it was only available for **Down Cutting**.

When machining parts with corners that were bigger than the tool insert, the tool would easily rapid into the stock. The user can now configure the Finish Turn cycle to avoid this type of collision.



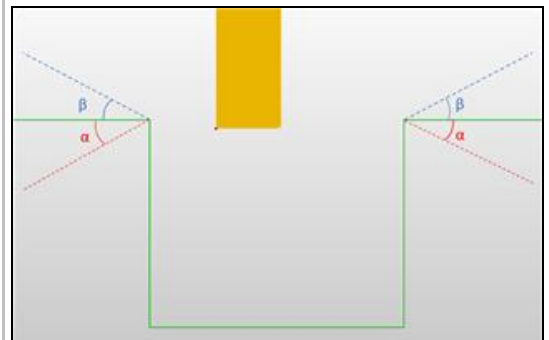
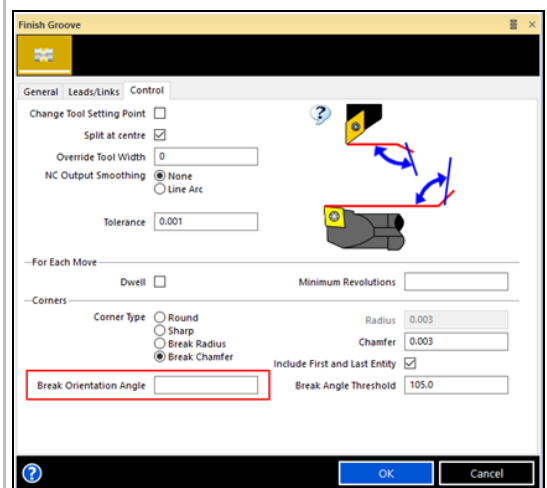
Finish Groove - Break Orientation Angle

For this release, a new **Break Orientation Angle** option has been added to the Control tab of the Finish Groove cycle. This functionality already exists for other turning cycles.

The functionality allows you to break the input geometry when a radius/chamfer is not provided. Values set in this field will be applied to the First Entity in the geometry as defined by the user when selecting the geometry.

Note: The option will not be available until **Include First and Last Entity** is selected.

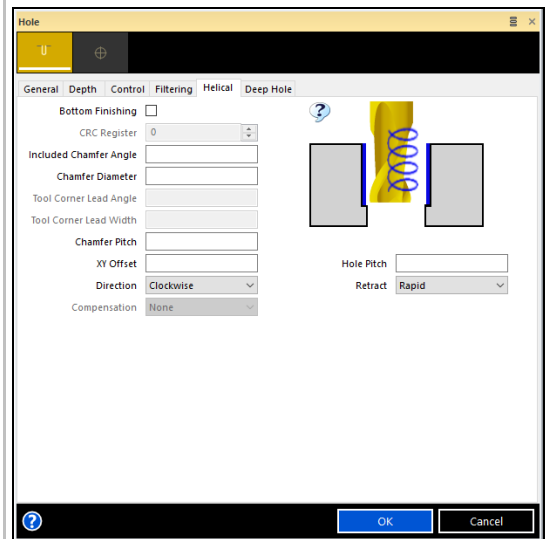
When generating the toolpath, the red angles in the image will be avoided so that the part is not damaged.



Hole Cycle - XY Offset added to Helical Hole

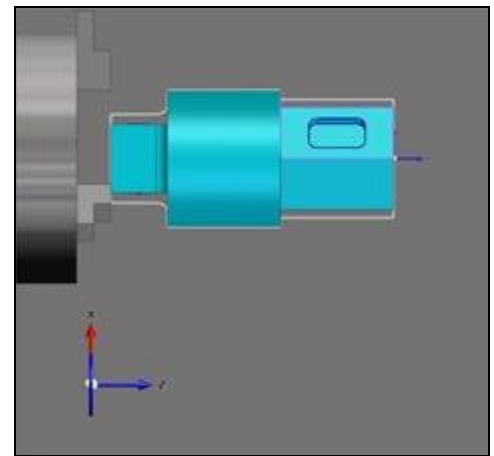
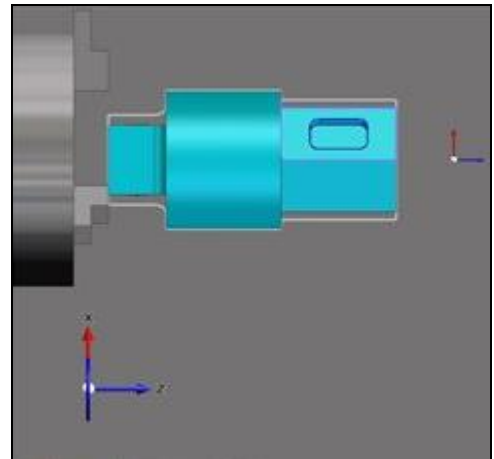
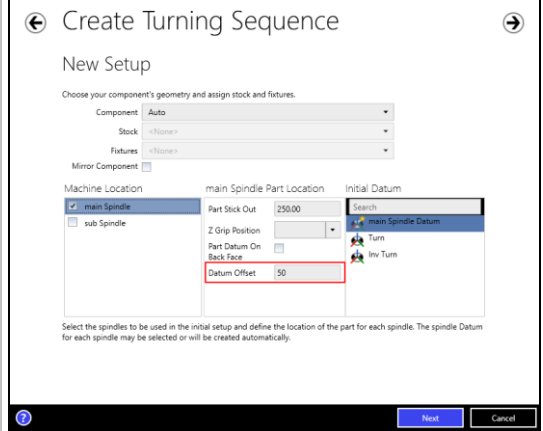
The Helical strategy of the Hole Cycle now has the ability to specify an **XY Offset**. This allows the cycle to leave material for either a secondary cycle to finish the hole or for a subsequent operation.

Note: There is no intention to add Z Offset; any offset at the bottom of the hole should be controlled with depth parameters.



Create Sequence - Ability to set Datum Offset for Part Datum

When creating a Turning Sequence, the user now has the ability to set an offset to the datum on either or both spindles using **Datum Offset**. When the offset is set, the Turn and Main Spindle do not need to be at the same location.



Advanced 5 Axis improvements

Support for Barrel Cutters

The EDGE CAM ToolStore and Tool Change command now supports Barrel cutters. This can be used by the Advanced 5 Axis cycle for the Surfaces and SWARF machining.

Side tilt by contact point

This feature provides a convenient, new tilting option for 5-axis machining. It allows you to define the tool axis tilting by specifying a contact point on the tool profile that is tangential to the machining surface along the toolpath. In addition to the contact point, a lead/lag angle can be specified to completely determine the orientation.

This feature significantly simplifies the operational setup, especially for tools with a complex profile geometry, e.g. barrel tools. The contact point is specified either by the axial distance or the distance along the tool profile starting on the tool tip. The user can specify a static contact point or a dynamic range, so that the contact point changes along the contour from the range start to the range end.

You can find this option under:

- Tool axis control Tab.
- Select - Tool axis will be tilted relative to cutting direction.
- Spherical tools should be defined.
- Set side tilt by the Contact point.

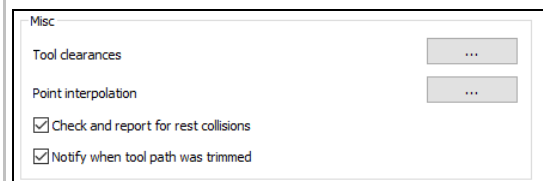
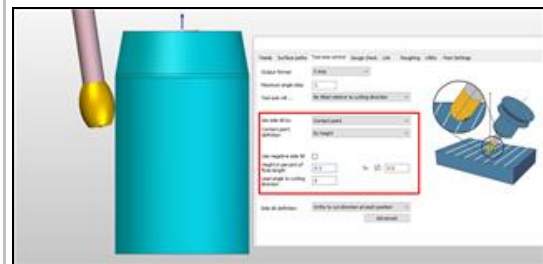
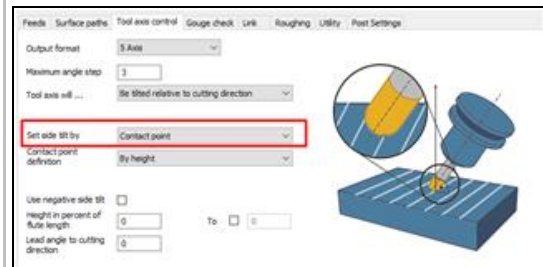
For example, a barrel cutter could set the contact point on the cutter to 0.3 to 0.5 of the flute height and, therefore, keep the cutting the range on the main radius of the cutter.

Existing toolpath - Notify if the toolpath was trimmed

When Strategy is Convert 3 to 5, a notification can be given when the toolpath was trimmed due to rest collisions.

Existing toolpath - check for rest collisions

When Strategy is Convert 3 to 5, a notification can be given of any remaining collisions.

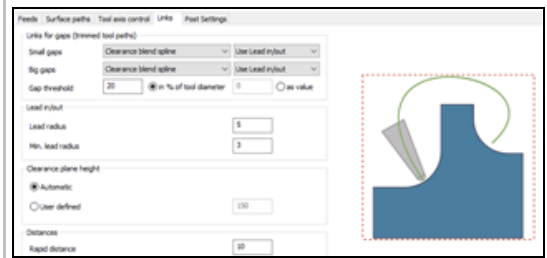


Existing Toolpath - Links

On Strategy Convert 3 to 5 axis, new links are created after trimming the toolpath whenever a collision between the tool and the workpiece can occur. The user can change the gap links settings for trimmed toolpaths to:

- Clearance area.
- Clearance blend spline.
- Rapid distance.

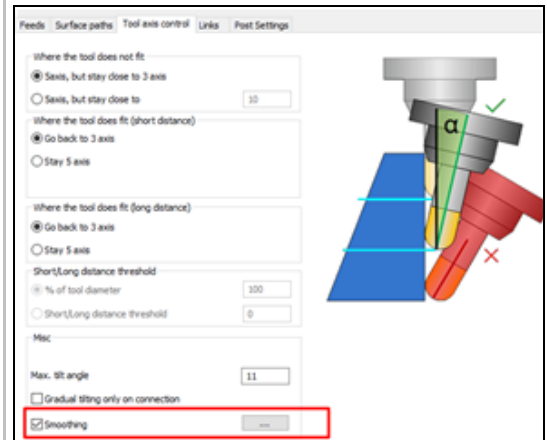
These types can be used together with the lead in/out option.



Existing toolpath - smoothing

Smoothing is an operation based on interpolation for optimising the contour while keeping the tilt angles within a limited range.

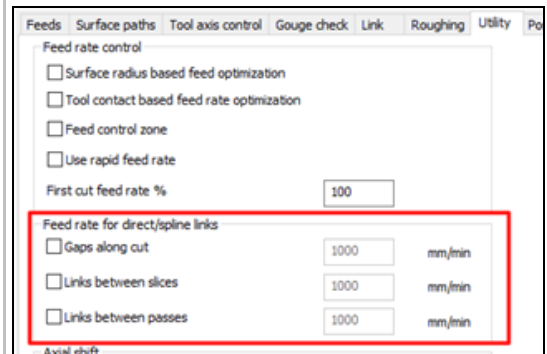
Smoothing by tilt angles tries to shape the converted toolpath (which may have some deviations) into a clearer one. This option modifies the path in which the contour's vectors are different to ensure that the tool moves fluently.



Feed rate for direct/spline links

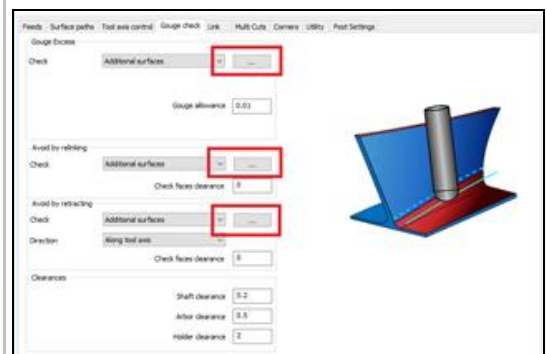
You can now control the feed rate for the type of links (direct or blend spline).

Available for Surface based strategies on the utility tab.



SWARF Gouge check - Additional surfaces

You now have the ability to add extra gouge checks for the SWARF machining strategy.



CAD Support Enhancements

Support for the latest CAD versions

The following CAD / file versions can now be loaded:

- Parasolid Library – Update to 31.1.188
- ACIS R1 - 2019 1.0
- Datakit Library – Upgrade to DTK-2019.2
- Solidworks 2019
- Inventor 2019
- Spaceclaim 2019
- Creo 5
- Granite - Creo 5 support
- NX 11 0 NX 12.0.0
- CATIA V5R8 – V5-6R2018.

Code Wizard Enhancements

Single Quadrant for Radial Arcs

Previously, Code Generator would only output full radial rotary arcs. However, as some machines cannot execute full radial rotary arcs, a new **Single Quadrant** option has been added to the **Rotary Axes** tab of the Machine Configuration dialog. Note that this option is only available when **Radial Arcs** is selected.

With the **Single Quadrant** option selected, the output now breaks arcs into quadrants.

A comparison of the NC code can be seen opposite.

Machine Parameters | Turnet | Mounting | Main Spindle | Main Spindle Gears | Spindle 2 | Spindle 2 Gears | Rotary Axes

☒ Radial Arcs

☒ Single Quadrant

Co-ordinate Output

☒ Wrapped at Specified Cycle Depth

☐ Wrapped at Wrap Radius

☐ As Original Unwrapped Data

test1-1.nc	test2-1.nc
:0001(CH1-ORIGINAL-TEST)	
(S) () (S001.CH1)	
G28 U0.0 V0.0	G28 U0.0 V0.0
M428	M428
G50 S5000	G50 S5000
G40 G80	G40 G80
N100 G54	N100 G54
G340T085085.(NC BOR D10.)	G340T085085.(NC BOR D10.)
G341T085.	G341T085.
G342 B0.0	G342 B0.0
M428	M428
G19	G19
M91	M91
S8000	S8000
M87	M87
G0 C0.0 M88	G0 C0.0 M88
G0 G98 G97 S8000	G0 G98 G97 S8000
G0 X260.0 Y0.0 Z-60.15 C0.0 M8	G0 X260.0 Y0.0 Z-60.15 C0.0 M8
X252.0	X252.0
G1 X244.0 F700.0	G1 X244.0 F700.0
Z10 W0 H0	Z10 W0 H0
G107 C125.0	G107 C125.0
G1 G41 X244.0 Z-75.15	G1 G41 X244.0 Z-75.15
G3 X244.0 Z-75.15 R25.15	G3 X244.0 Z-50.0 C11.528 R25.15
G1 G40 X244.0 Z-60.15	G3 X244.0 Z-24.85 C0.0 R25.15
G107 C0.0	G3 X244.0 Z-50.0 C-11.528 R25.15
G1 X238.0	G3 X244.0 Z-75.15 C0.0 R25.15
G19 W0 H0	G1 G40 X244.0 Z-60.15
G107 C125.0	G107 C0.0
G1 G41 X238.0 Z-75.15	G1 X238.0
G3 X238.0 Z-75.15 R25.15	G19 W0 H0
G1 G40 X238.0 Z-60.15	G107 C125.0

Code Generator Enhancements

System Variables for Gun Drilling

Two new system variables (DEEPHOLE and ENTRYDEPTH) have been added to the following Code Generator macros:

- Milling Macro Reference (MACRO 9 - DRILL CYCLE).
- Turning Macro Reference (MACRO 9 - DRILL CYCLE).

ToolStore Enhancements

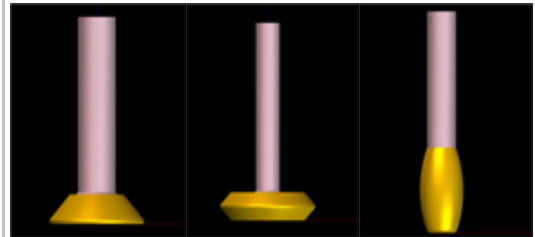
EDGE CAM ToolStore - New Milling Tool Types

Three Milling Tool Types have been added to the ToolStore:

- Barrel Cutters for 5 Axis Machining.
- DoveTail and Double Angle Tools for Chamfer generation.

For Dovetail and Double Angle tools, you can specify a **Diameter**, **Included Angle** and **Flute Length**. Note that the **Included Angle** setting in the ToolStore is named **Angle** in the EDGE CAM Tooling dialogs. Optionally, specify a **Corner Radius**.

For Barrel tools, you can specify an **Upper Diameter**, **Lower Diameter**, **Flute Length** and **Profile Radius** or specify a **Max Diameter** from which the **Profile Radius** can be derived. Optionally, specify a **Corner Radius**.



Notes		Technology		Additional	
General	Geometry	Mounting	Angled Head	Allocation	
Diameter	<input type="text"/>	<input type="checkbox"/> Centre Cut			
Corner Radius	<input type="text"/>	Through Coolant	Off		
Flute Length	<input type="text"/>	Max Plunge Depth	<input type="text"/>		
Teeth/Flutes	<input type="text"/>	Ramp Angle	<input type="text"/>		
Hand Of Tool		Included Angle	<input type="text"/>		
<input type="radio"/> Left Hand <input type="radio"/> Right Hand		Undercut Distance	<input type="text"/>		
Shank					
Length		<input type="text"/>			
Diameter		<input type="text"/>			
<input checked="" type="checkbox"/> Visible					
Graphic <input type="text"/>					

Notes		Technology		Additional	
General	Geometry	Mounting	Angled Head	Allocation	
Lower Diameter	<input type="text"/>	Upper Diameter	<input type="text"/>		
Corner Radius	<input type="text"/>	Max Diameter	<input type="text"/>		
Flute Length	<input type="text"/>	Profile Radius	<input type="text"/>		
Hand Of Tool		<input type="checkbox"/> Centre Cut			
<input type="radio"/> Left Hand <input type="radio"/> Right Hand		Through Coolant	Off		
Shank		Max Plunge Depth	<input type="text"/>		
Length		Ramp Angle	<input type="text"/>		
Diameter		Undercut Distance	<input type="text"/>		
<input checked="" type="checkbox"/> Visible		Teeth/Flutes	<input type="text"/>		
Graphic <input type="text"/>					

Important Licensing Changes

License Wizard - Configure Network licenses

The License Wizard has been enhanced to enable users to configure network licenses after selecting a network license server. Previously, the user could only select the server and would then have to run License Manager to Configure Network licenses:

- Selecting the **Save** button saves a license configuration, not a Profile.
- Selecting the **Save As** button allows the user to save the license configuration as a Profile.

Other improvements to License Wizard

The License Wizard has been enhanced to enable users to configure network licenses after selecting a network license server. Previously, the user could only select the server and would then have to run License Manager to Configure Network licenses:

- The brand name and the name of the selected network license server are displayed in the Title Bar.
- The new **Server** button allows the user to switch between different network license servers.
- The check box to enable profile selection on startup is included in the License Wizard.

Note: Network licenses and profiles can still be configured in License Manager.