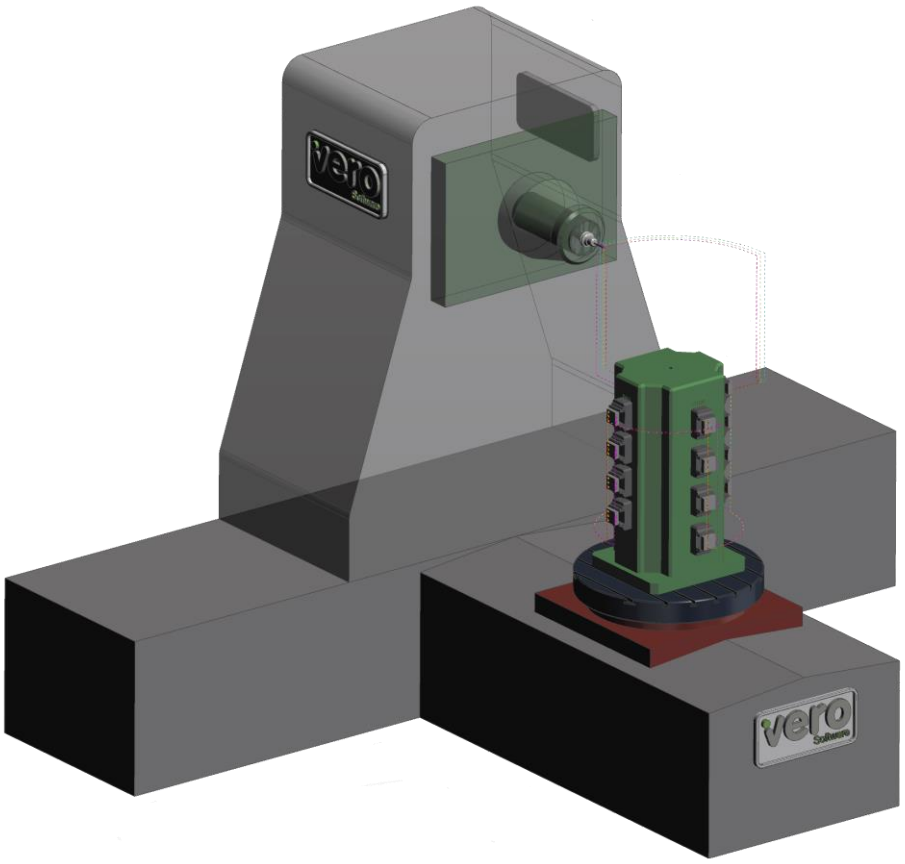
What's New in Edgecam 2018 R2



This document highlights new product features and enhancements in Edgecam 2018 R2.

To run Edgecam and Part Modeler 2018 R2, the expiry date in the license must be April 2018 or later.

Contents

Tombstone Enhancements

Wire Enhancements

Simulator Enhancements

Code Generator Enhancements

ToolStore Enhancements

Important Licensing Changes

‘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 2018 R1.

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

Important Information

CLS Licensing Changes

Please see Important Licensing Changes.

Adveon

Sandvik are withdrawing support for Adveon as it has been superseded by CoroPlus ToolLibrary. 2018 R2 will not support Adveon. You can still use 2018 R1 to extract a solid model from Adveon and link the data to tools in 2018 R2. We intend to implement CoroPlus in a future version of Edgecam.

Autodesk Vault

Autodesk Vault was removed from the price list a few years ago. We are now announcing retirement of this module from 2019 R1.

Retirement of IGES Healing

A business decision has been made to retire the Cset Ci2x IGES loader used in Edgecam and Part Modeler.

Note: This will not come into immediate effect and will happen over the next few releases of the software.

To replace the Cset Ci2x IGES loader we have started the initial work to further develop SolidLink which will enable geometry (lines, arcs, etc) and surfaces to be extracted from IGES files that contain such elements.

Designer

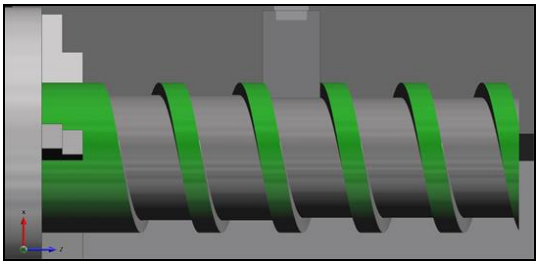
Edgecam Designer 2018 R2 and will be listed on the DVD StartHere menu as a download.

C/C++ - Continued support

We made a statement in previous releases that the support for C/C++ PDI's would be retired. We have decided to continue current support for at least 2019 R2. We still strongly recommend that no new development should be undertaken using PDI and that you move to .NET Plugin.

Manufacture Enhancements

Thread Turning cycle is now available for all Turn Tool Types

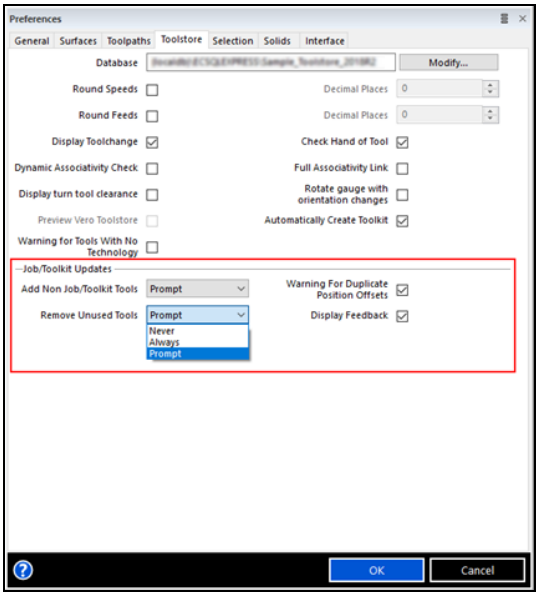


Previously, in Edgecam, Thread Turning was only possible with a Threading tool.

However, as thread forms come in all shapes and sizes, we now allow the cycle to be used with any Turn Tool Type.

The example shows a square thread being formed with a grooving tool.

Preferences added to control behaviour of Update Job/Toolkit from Edgecam

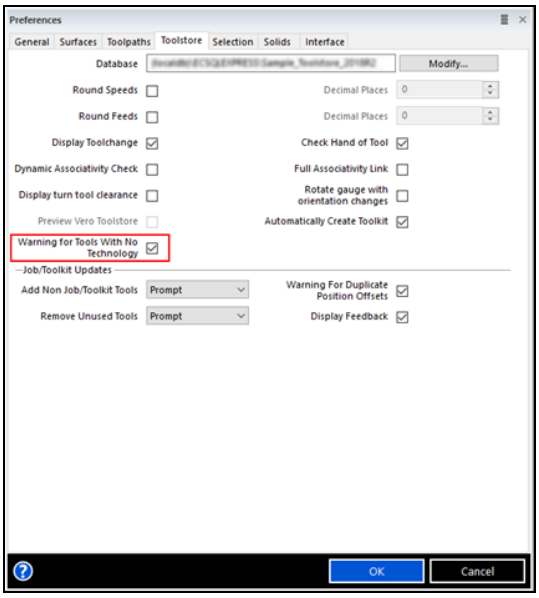


Preferences have been added to the **Toolstore** tab of the **Preferences** dialog.

These 'Job/Toolkit Updates' preferences enable the user to control the behaviour when updating a Job/Toolkit from Edgecam.

The default behaviour is to prompt each time tools are added or removed from the Job/Toolkit, but the user can now control this behaviour with the four preferences which have been added.

Preference added to output a warning when selecting tools without an associated technology



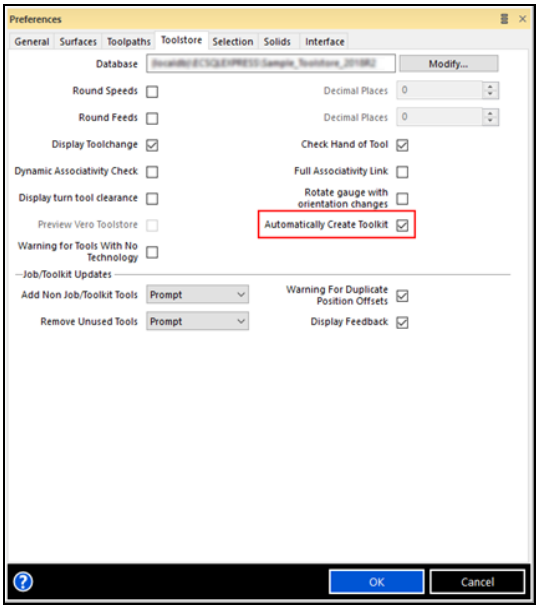
A Preference has been added to the **Toolstore** tab of the **Preferences** dialog to output a warning when selecting tools without an associated technology.

The user needs to be informed when the selected tool does not have an associated technology so that they can manually set the speed, feed and depth of cut values for such tools:

- With **Warning for Tools With No Technology** selected, a check is made and a warning popup is displayed when loading the tool.
- The warning is also written to the feedback window irrespective of whether the option is selected.
- If the user edits the setup and changes the part material, some tools in the sequence may no longer have an associated technology; therefore, upon sequence regeneration, warnings are written to the feedback window to inform the user of these tools. Users are then expected to manually update any affected cycles with appropriate speed, feed and depth of cut values.

Note: During regeneration, the warning is only written for tools which have associative checked.

Option to not create a new Toolkit automatically

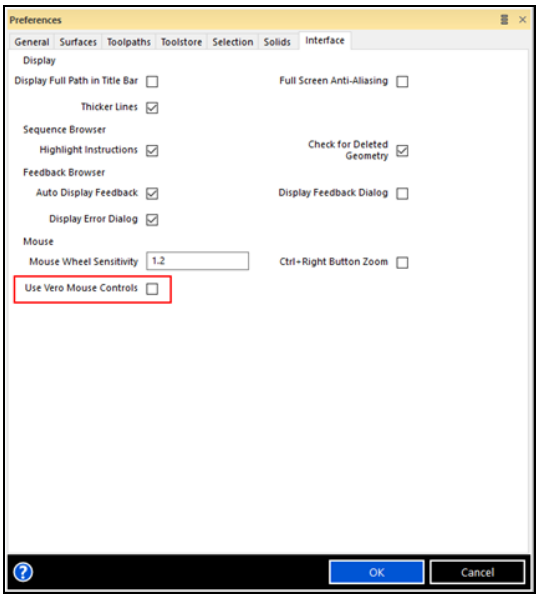


Previously, **Create Sequence** created toolkits every time that users created a sequence which could cause issues if they did not use ToolStore.

For this release a new **Automatically Create Toolkit** preference has been added to the **Toolstore** tab of the **Preferences** dialog which defaults to on. If this option is unchecked, when the post is selected, Edgecam will not create a new Toolkit by default.

Note: To create a new Toolkit, type a name in the Toolkit field or click on one of the existing Toolkits to make a copy.

New Mouse Controls



For this release a new **Use Vero Mouse Controls** preference has been added to the **Interface** tab of the **Preferences** dialog.

Check this to use the mouse controls which are shared with other Vero products. The differences for Edgecam users are highlighted in the following list:

Mouse controls

Pan	Hold Left & Right mouse buttons and hover.
Zoom in	Scroll wheel forward (on cursor position).
Zoom out	Scroll wheel backward (on cursor position).
Zoom extend	Double-click scroll wheel.
Orbit (rotate)	Hold Right mouse button and hover (screen centre).
Orbit (rotate on point)	CTRL + Right mouse button and hover (defined point).

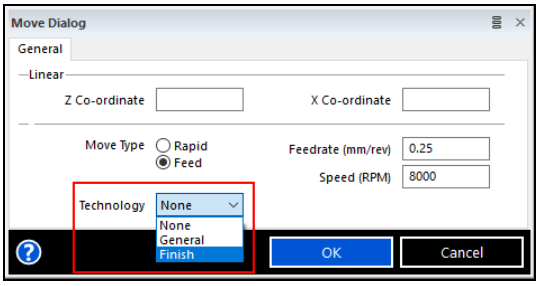
Graphical aids

Fast zoom in	Spacebar (release space bar returns to original magnification).
Retain fast zoom in	ALT + spacebar.

Selection (picking) methods

SHIFT + Left mouse button on a element	Allows the colour of the selected element to be set as a filter to pick all the other elements of the same colour.
ALT + Left mouse button on a element	Allows the system to chain all of the tangent elements in a single selection.
CTRL + Left mouse button on an element	Allows an entity to be added or removed from a group selection.
Window selection left to right	Selects all of the elements fully included in the drawn box.
Window selection right to left	Selects all of the elements fully included in the drawn box and intersected by the box edges.

Move Dialog command now includes the Technology modifier



The **Technology** drop-down modifier has been added to the **Move Dialog** command:

- Active when **Move Type** is set to **Feed**.
- Enables the technology parameters associated with the tool to populate the **Speed** and **Feed** fields.

Edgecam Solid Machinist for Creo license added

In Edgecam 2018 R2, we are introducing a new license module - Solid Machinist for Creo.

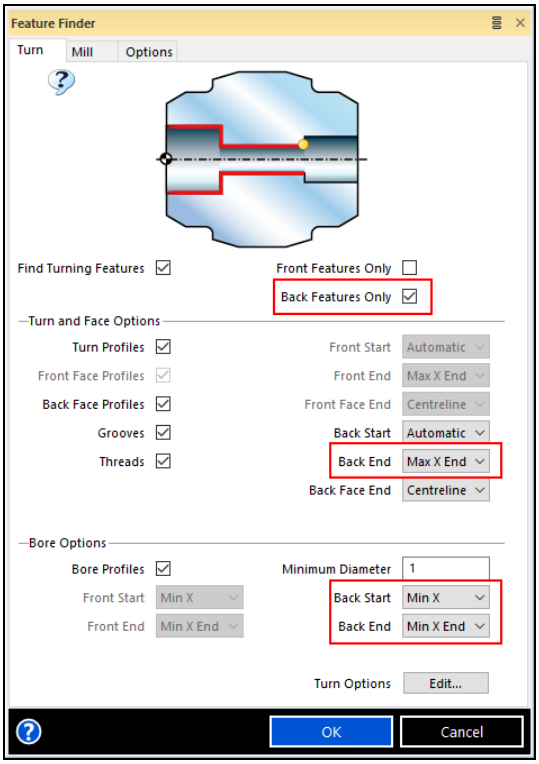
Previously, in order to load Creo parts, the customers needed the Solid Machinist for Granite module - which allows Edgecam to use the PTC Granite solid kernel to load the solids.

The new Solid Machinist for Creo uses the Parasolid kernel, therefore Edgecam will load the solids as Parasolid models.

Note:

Resellers and other users with both Solid Machinist for Creo AND Granite licenses available will load Creo CAD files as Granite models as before and save/load these in PPFs but the PCI Variable !LoadCreoAsParasolid with a Numeric Value of 1 should allow loading of Creo models as Parasolid and save those models in PPFs.

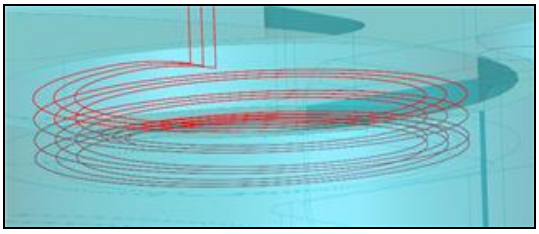
Feature Find Back Turn Features Only



For this release, we have added a new **Back Features Only** option to Turn Feature Find. This can be useful on sub spindle setups on which only back turn features are required:

- New options have been added to control where Back features start and end.
- To find both Front and Back features, uncheck the **Front Features Only** and **Back Features Only** check boxes.

Profiling - Multiple XY Offset Passes with Helical

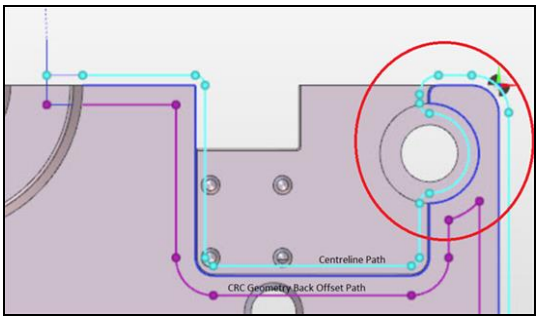


For this release, we have implemented the option to select **Helical** on multiple xy offset passes:

- At this stage, support is for vertical walls only that have exact geometry.
- If the input geometry is unsuitable for Multi Pass Helical, a warning is displayed and the toolpath generated will be planar.

Note: In previous versions, the **Helical** option on the Depth tab was unavailable when **Multiple Passes** were set on the General tab.

Profiling - Disallowing Profiling with a Taper tool with CRC=Geometry



The combination of Profiling with a Taper tool with CRC **Compensation** = **Geometry** on certain geometries can lead to the back offset path intersecting with itself resulting in the back offset path having a different number of elements to the centreline path as shown in the image.

In this situation, the machine would follow the back offset path and machine straight through the part.

Simulator simulates the centreline path and, therefore, this situation can be easily missed.

We are, therefore, disallowing CRC **Compensation** = **Geometry** when Profiling with a Taper tool.

Upon detecting this combination, a screen message will be displayed stating that CRC **Compensation** = **Geometry** is not a valid option for taper tools and the user will be asked to select either CRC **Compensation** = **None** or CRC **Compensation** = **Centre Line**; in this way, the situation cannot be missed.

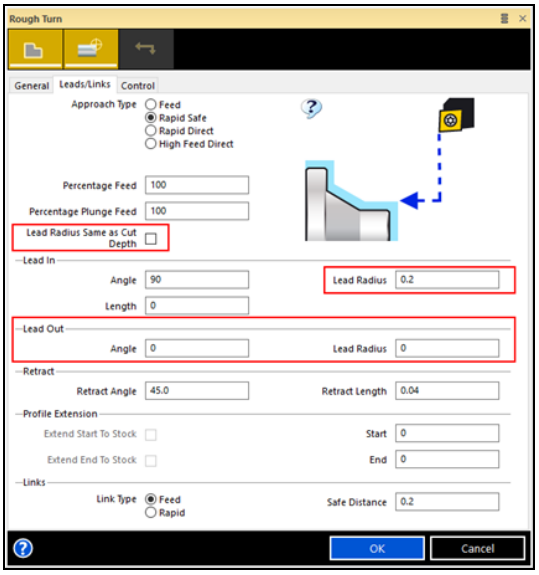
Profiling - Solid Faces Performance Improvement

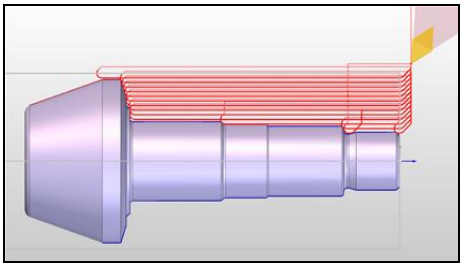
The Profiling cycle has been improved to optimise the data which it uses to calculate toolpaths when the user is picking faces of a solid model.

The previous and usual way would be to collect the whole model geometry so that the cycle could safely compute links that would not gouge the model outside of the selected faces.

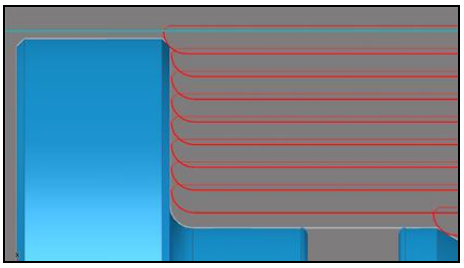
This method has been reviewed and changes made to reduce the amount of data collected and used for calculation. Links will still be safe and cycle calculation should be considerably faster, especially on large parts (models).

Rough Turn - Added Arc Lead On/Off

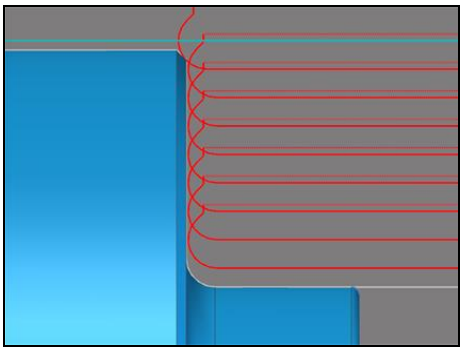




Example Lead Out Radius with Rough Cuts Only:



Example of Lead In Radius with Rough Cuts Only - Reverse Cut Direction:



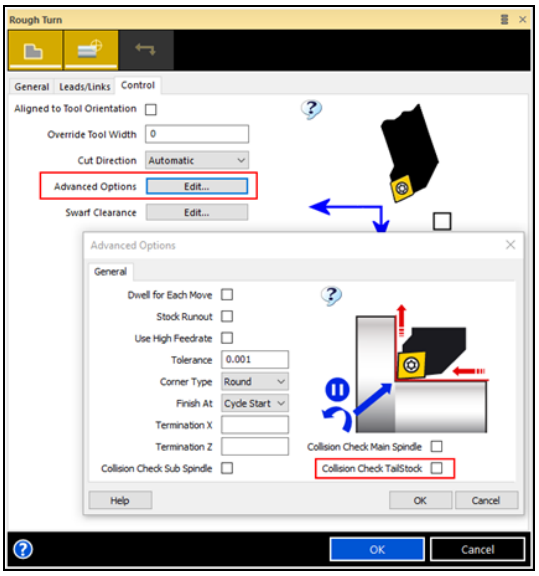
In the Rough Turn cycle, it is now possible to roll on and off each cut with a tangential arc:

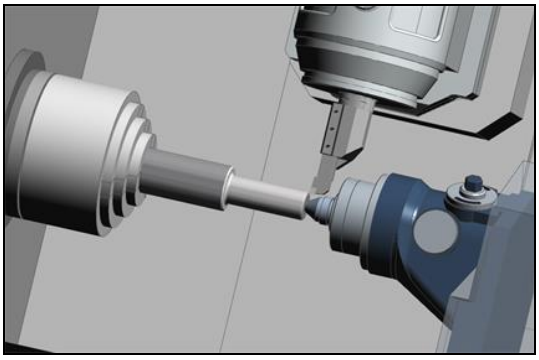
- This is the preferred method for entering hard materials such as Inconel, titanium etc.
- It is also required when the direction of the cut is reversed, i.e. cutting back to front when leading onto the face. This is the preferred cut direction for certain types of Inserts.
- The feedrate of the arc lead on can be adjusted using the **Percentage Plunge Feed** option.

The following modifiers have been added:

- **Lead Radius Same as Cut Depth** - Checking this greys out the **Lead Radius** option and sets it to the same value as the **Cut Increment**.
- **Lead In / Out Radius** - Used in conjunction with the Lead **Angle** to control the size and amount of arc lead.

Rough Turn - Collision Avoidance for Tailstocks





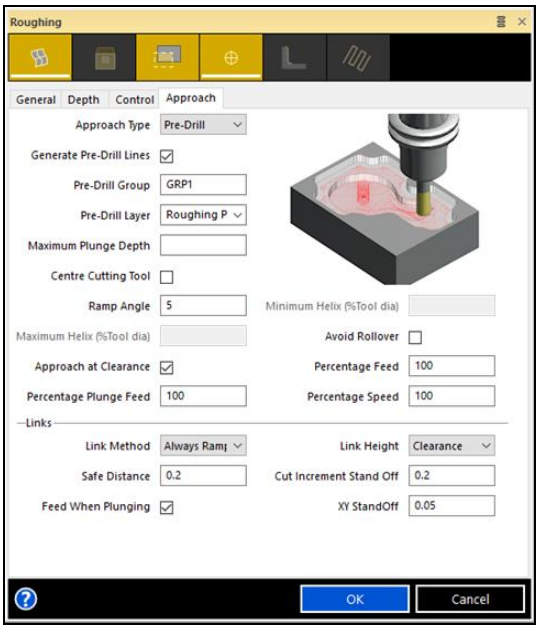
The Rough Turn cycle has been enhanced to collision check the Tailstock(s).

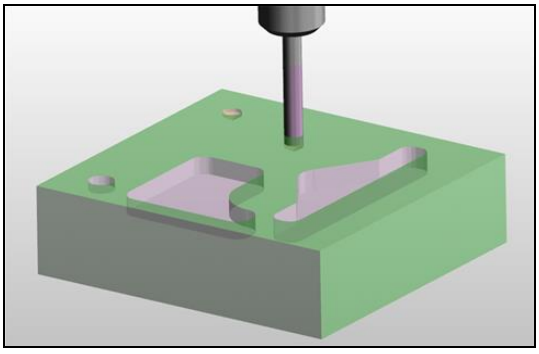
A **Collision Check Tailstock** option has been added to the Advanced Options on the Control tab which prevents the tool colliding with the Tailstock:

- The **Fixture Offset** in Update Fixtures (General tab) to determine the minimum allowable clearance distance between tool and Tailstock.
- In Edgecam, the Tailstock is controlled using the **Move Tailstock** command.
- The Rough Turn toolpath is trimmed to the Tailstock boundary + any fixture offset.

Note: The Tailstock needs to be defined in Code Wizard as part of the machine; Tailstocks which are just fixtures are not supported.

Roughing Cycle - Plunge Approach and Pre-drill Hole Locations





The Roughing cycle has been enhanced to be able to generate the pre-drill positions and plunge at these locations.

With **Approach Type** set to **Pre-Drill** and **Generate Pre-Drill Lines** selected, the Roughing cycle will plunge at the pre-drill locations assuming that the holes have already been drilled. A **Safe Distance** value must be specified to avoid making a rapid move into the rest material left by the drill tip angle.

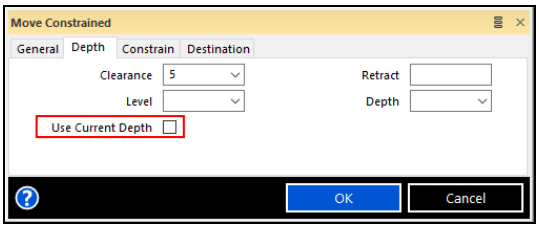
After the Roughing cycle has generated the pre-drill group of lines, the user can insert a Hole cycle prior to the Roughing to drill these; selecting a drill with a slightly bigger diameter than that used for roughing. In the Hole cycle:

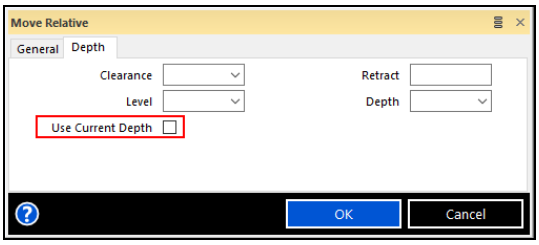
- Select **Vertical Lines** on the Filtering tab.
- Check **Multi Level** and **Multi Depth** on the Depth tab; the level and depth are taken from each line element in the group;
- Set a value of zero for the **Level** and **Depth**.

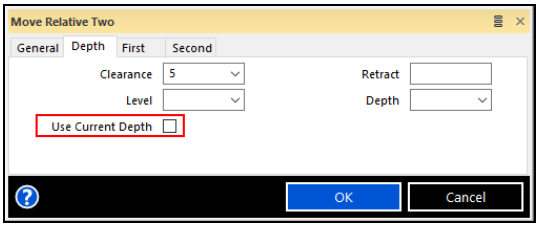
Note:

- The pre-drill group can change if the Roughing cycle is modified; a message will be displayed advising that any associated pre drill hole cycles need to be regenerated to remain associative to the roughing.
- Multiple Pre-Drill groups on the same CPL can be drilled by the one cycle.
- A separate hole cycle is needed to drill groups on multiple CPLs as the hole cycle is not able to automatically index when the input geometry is wireframe.

Move Constrained, Relative and Relative Two - Option added to Perform Moves at Current Depth







For this release, we have added a **Use Current Depth** check box to the Move Constrained, Relative and Relative Two commands which, when checked, performs moves at the current tool depth:

- In previous versions, to perform these moves at current depth, the **Depth** value had to be set on the dialog but, in some circumstances, this value was not known or easily determined.
- If the first move specifies the **Depth** and the following commands set **Use Current Depth** then they will be updated if the first is edited.

Transforming Instructions on Lathes with more than Two Turrets

On a Lathe with more than two turrets (i.e two upper and two lower), when selecting instructions to Transform, we now list the instructions for the active turret only.

Previously, the instructions for two turrets were listed but this meant that instructions for the other two turrets could not be selected for transformations.

Surface Machining Cycles are now available In Rotary Mode with an Axial Mounted Tool

The following Surface Machining cycles are now available in the Turning environment when in Rotary mode with an axial mounted tool:

- Parallel Lace.
- Constant Cup Finishing.
- Rest finishing.
- Flow surface.
- Pencil Mill.
- Project Toolpath.
- Project Boundary Collapse.
- Project Flow Curves.
- Project Circular Pattern.

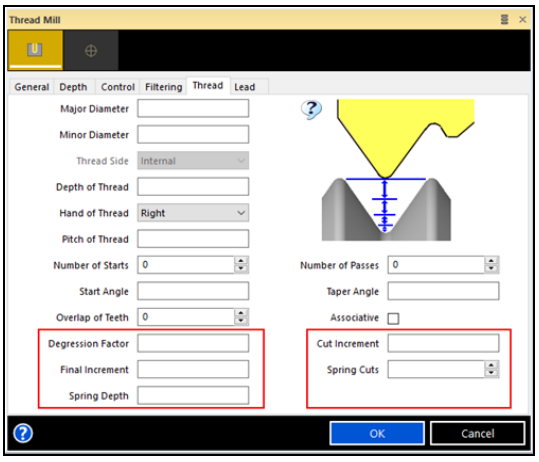
The cycles are also available in the Milling environment in Rotary mode when the rotary axis is parallel to the tool axis.

Note: To use the Surface Machining cycles in Rotary Mode in Milling, the **Force Planar in Rotary Axial Milling** option in the **Machine Parameters** dialog of Code Wizard (**Configure > Machine Parameters**) must be unchecked.

Thread Milling

The Thread Mill cycle has been enhanced and now supports better control for multi-passes allowing variable depth. It also provides greater control over leads and between-passes links.

Variable depths for multi-passes - Spring passes



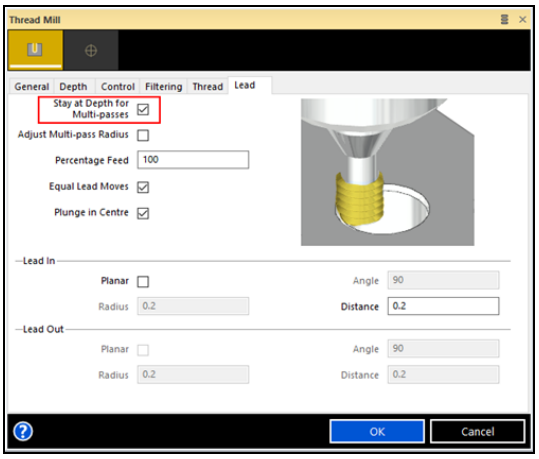
The following options have been added to the Thread tab:

- **Degression Factor** - A factor that progressively reduces the depth of each cut, as the number of cuts increases. Must be greater than 1 (need not be a whole number).
- **Cut Increment** - Specifies the distance that the tool moves in the cut direction with each pass. If the cut factor is not 1.0 then the **Cut Increment** represents only the first depth of cut as the toolpath will degress in depth.
- **Final Increment** - Specifies the last cut increment. Actual cut increments for each pass are interpolated from the **Cut Increment** and the **Final Increment**.
- **Spring Cuts** - Specifies the number of finishing passes made using **Spring Depth** to the Total Depth. This allows for any "spring" in the tool or workpiece.
- **Spring Depth** - Specifies the depth of material to be removed by each spring cut.

Notes:

- **Number of Passes** and **Cut Increment** are mutually exclusive.
- **Degression Factor** and **Final Increment** are mutually exclusive.
- **Degression Factor** can only be used if there are multiple passes.
- **Spring Cuts** can only be created if there is more than one pass.

Link between multi-passes

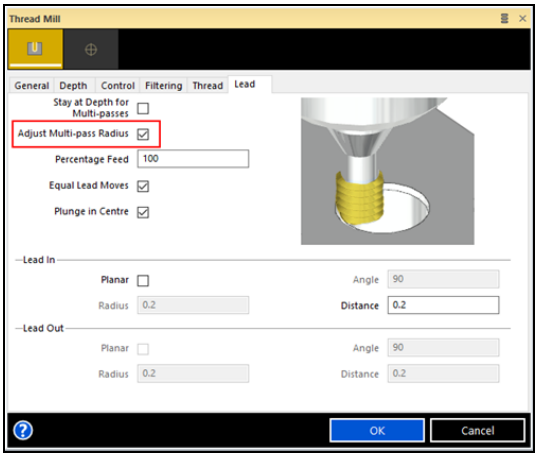


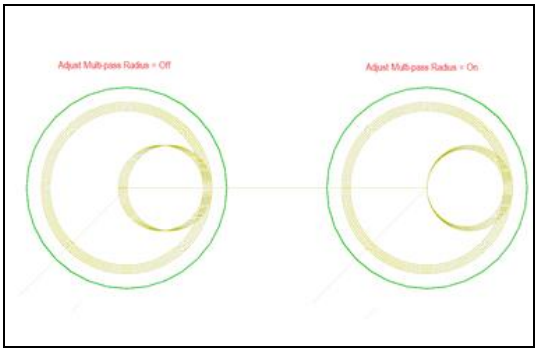
Previously, on an internal thread, when there were multiple passes on a thread, the tool would usually retract back to Retract height and then plunge again to continue with the next pass. This could unnecessarily increase cycle time.

A **Stay at Depth for Multi-passes** option has been added to optionally maintain the tool at depth, forcing it to make a straight move to the next pass start point.

Note: Only available on internal threads.

Leads - Defaults and variable radius





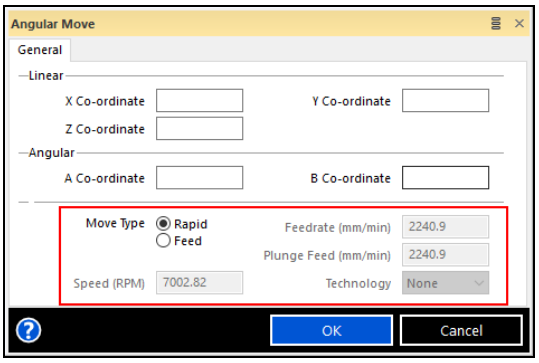
The calculation for the default lead radius on internal threads has been slightly changed in response to user feedback.

In this release, for an Internal Thread, when **Plunge in Centre** is enabled and the user has not specified a lead radius, the default will be to make the radius in such a way that a 180 degree radius would start exactly on the centre of the thread.

Additionally, when the thread contains multiple passes, an **Adjust Multi-pass Radius** option to vary the radius been implemented. This will vary the radius for each pass so that the start point is maintained constant.

Note: Only available for internal threads.

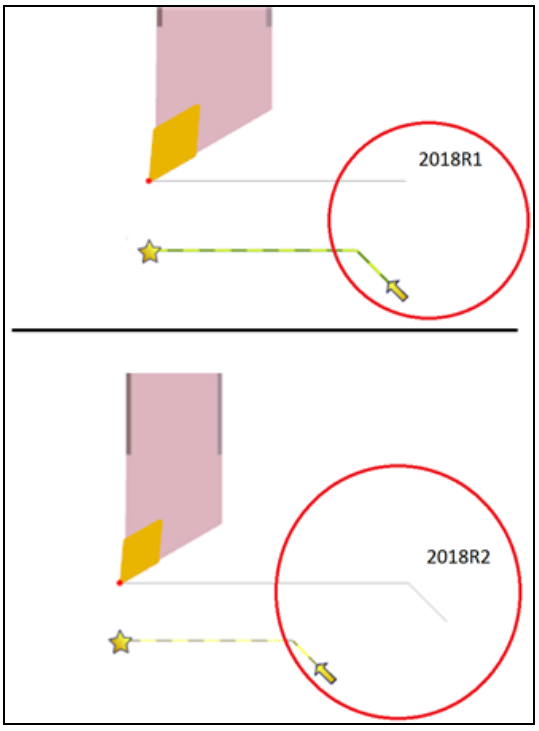
Move Angular Rapids disables Technology modifiers



In previous versions, a Move Angular in Rapid Mode (**Move Type** set to **Rapid**) could override Technology values for Speed and Feed which resulted in incorrect values in the NC for the next machining cycle.

These modifiers are now unavailable when Rapid Mode is set and the NEXTSPEED look ahead in the code generator has been fixed to ensure that correct values are output to the NC.

Change to Turning Cycles XZ Offset



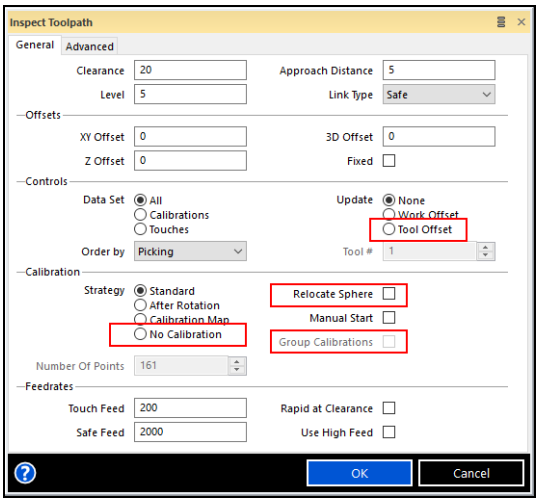
For this release, we have made a change to the Turning Cycles XZ Offset to overcome an issue with toolpath segments being shortened or even being removed in some cases. This was especially noticeable when large XZ offsets were specified as shown in the image.

Note: This change affects all turning cycles which use XZ offset.

Edgecam Inspection improvements

As part of the ongoing improvements to Edgecam Inspection, a number of enhancements have been implemented.

Inspect Toolpath - General tab



Calibration strategy – Group by axis

The **Group Calibrations** option allows the user to calibrate all measuring vectors at the indexed position before moving to measurement.

More features for Tool Offset

Plane and Arc features were added to the features that can be picked to update Tool Offset.

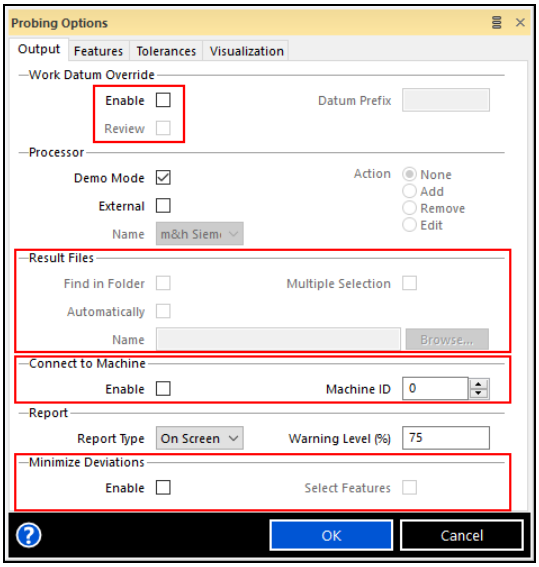
Enable Relocate Sphere for all calibration strategies

The **Re-Calibrate** option has been renamed to **Relocate Sphere**. This option is now available for all calibration strategies.

Calibration strategies – No Calibration

The **No Calibration** strategy has been added. When selected, no calibration procedure will be created and a calibration file to evaluate results will not be required.

Probing Options - Output tab



Work Datum Override

With this new command, the user can specify the work offset numbers for datums used in the sequence.

Recalculation of deviations with BestFit (Workpiece alignment)

The **Minimize Deviations** option enables the user to use the Best Fit function for minimizing errors in deviations by aligning real points with nominal data.

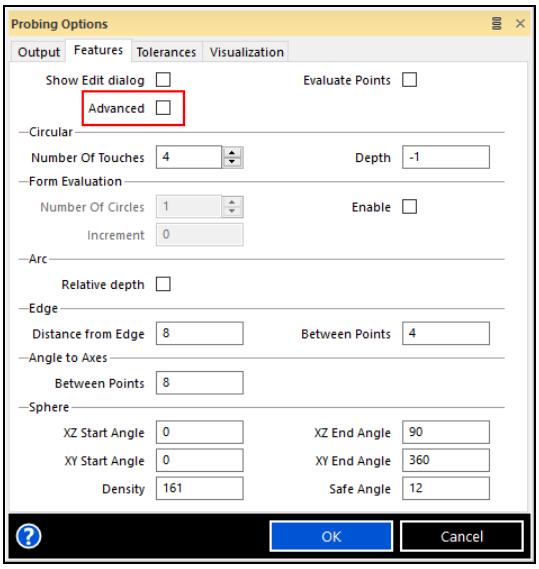
Watcher to process results automatically

The **Result Files** options have been enhanced to automatically process result files when they appear in a specified folder.

Automatically send NC to machine (CNC Gateway)

The **Connect to Machine** options enable NC programs to be automatically sent to the machine via CNC Gateway.

Probing Options - Features tab



Advanced Options for Feature tolerances

Some feature types now have an **Advanced** tab with options for values (deviations) which can be excluded from the report. Select the **Advanced** option to display the Advanced tab for these feature types.

Other Improvements

- **Rectangle feature**

  The Rectangle feature has been implemented. To create this feature, the user simply needs to pick two perpendicular faces.

- **Function to check journal file for completion**

  The Heidenhain controller does not have a function to delete a file and, therefore, continues writing to the same file. If the session has failed, the result file may have multiple data sets, some of which are incomplete. Therefore, in this release, incomplete cycles are ignored and a journal file with a fragment of the resulting data plus complete resulting data will be processed.

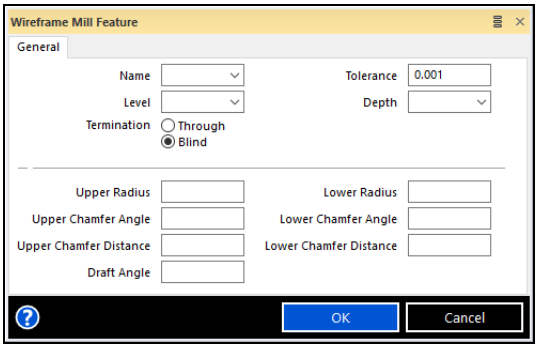
- **Read result files with floating decimal point**

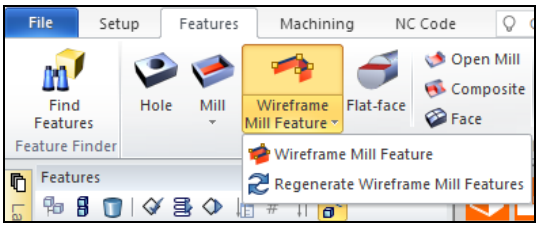
  Some controllers output values smaller than 0.01 micron as a number with floating decimal points, for example, 0.565678EX-7. In this release, the value will be detected and treated as 0.

- **Inspection Cycle with Protected move**

  In this release, different feedrates can be set for links below Clearance/Level and probe touch moves.

Mill Features from Wireframe





We have added a new command to enhance Feature creation capabilities which now creates Milling Features from Wireframe.

This command can be used when the faces or edges of a solid are not suitable to create the feature. The geometric input to the feature is always a 2D wireframe profile comprised of lines, arcs, etc:

- The level and depth of the feature are specified on the dialog. Additional modifiers are available for defining an upper/lower radius or chamfer and draft wall angle.
- When the Feature is edited, additional parameters will be available allowing you to change the Coordinate Input, Reverse the Feature side and to add a Comment to the Feature.
- It is also possible to give the feature a name to ensure that it is recognisable in the feature browser window.

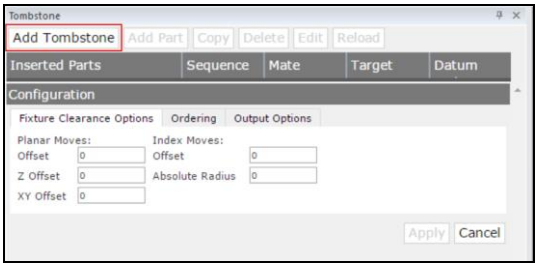
Due to the associativity between the wireframe and the feature created from it, if you have a feature created from a rectangle (closed shape), removing one of the lines will make the new feature an open shape. However, the regeneration is not automatic to avoid issues with features changing their geometry without user interaction and must be performed manually using the **Regenerate Wireframe Mill Features** command.

Notes:

- Associativity is limited to simple changes to the input geometry.
- If the wireframe geometry is extracted from a solid model, there will be no associativity between the Solid and the Wireframe.
- Wireframe Mill features are designed for use with solid models and are, therefore, only available when a solid is present in the Edgecam session. When a solid model is present, Wireframe Mill features can be created and machined using cycles or strategies. These features are not designed for wireframe only systems.

Tombstone Enhancements

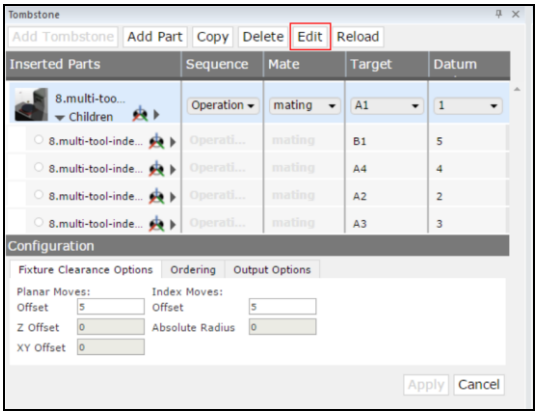
Add the Tombstone 'Seed' Part



Previously, the Tombstone method required the user to manually open the 'seed' file and then insert parts on it to create the Tombstone assembly. Once finished, it was the responsibility of the user to correctly save that job with the possibility that the 'seed' file could be lost which could have been extremely time consuming.

In this release, the module has been enhanced and the first step is now to insert the Tombstone ('seed') file. Only after this is done will the user be able to continue with the assembly. This also forces a **Save as** which prevents the accidental overwriting of the precious 'seed' file.

Launch Inserted Parts for Editing

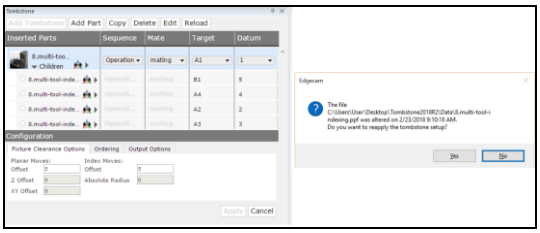


Once the user has finished the assembly, and applied Tombstone, the module now allows parts to be edited from inside the browser. This will launch a new session of Edgecam for that specific part enabling the user to edit as required.

Note:

- The part needs to be available in the original location (at the time of insertion into Tombstone), otherwise, it cannot be edited.
- Children cannot be edited because they are instances of a Parent part which can be edited.
- To edit a part, select the Parent in the Browser which then enables the **Edit** button.

Reload Mechanism



Tombstone now maintains a live link to each part that was inserted into its assembly. That link remains valid while the location does not change. When it detects that a part has been changed (by checking its time stamp), it will prompt the user to either reconstruct the assembly with the new edited part or maintain the current state.

If **Yes** is selected, Tombstone will recreate the assembly. Note that if a part cannot be found, Tombstone will warn the user and prevent the rebuild.

Editing of the Tombstone Assembly

When a Tombstone sequence already exists, the user may want to change, for example, the assembly, quantity, clearances, ordering or output. When a change is made on the browser, the **Apply** button is reactivated and the user will be prompted depending on the level of editing:

- If Datums, Targets or Mating are changed, the entire assembly needs to be reconstructed. The user will be prompted perform the regeneration.
- If Offset, Output or Ordering are changed, the Tombstone toolpath needs to be regenerated but the assembly does not require reconstruction. The user will be warned to regenerate the sequence which will then update the toolpaths.

Performance

Tombstone is considerably faster to apply. The engine has been enhanced to reduce unnecessary calculations by being smarter in reducing data and calculation for child parts. The reduction in processing time can be up to 80% depending on the complexity of the assembly.

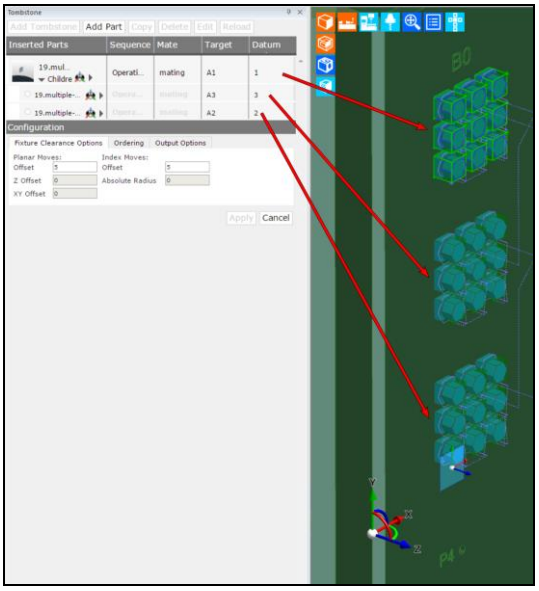
Improved Ordering - Closest Next

The toolpath and rationalisation algorithm has been enhanced to improve the order of toolpaths and ensure that, whenever possible, the closest next part is machined. Previously, Tombstone would follow the order in which parts were inserted which could, potentially, make each job a few seconds longer. In a large scale production environment, this enhancement should make a considerable difference.

Unlimited Number of Parts to Insert

Previously, Tombstone was limited to two distinctive parts being added to its assembly. This limit has been removed and now the user can insert as many parts as required.

Parts with Multiple Components



Parts which contain multiple components (copies or distinctive parts) are now supported on Tombstone.

It is important that the component structure on the insert-part is correctly defined, otherwise, the elements might not be correctly recognised.

Subroutines for Holes

Previously, subroutines for Hole cycles have been disabled due to conflicts between Tombstone subroutines and hole points subroutines.

This issue has been resolved and Hole cycle points will now be grouped into Subroutines when applicable.

Note: At this stage, Tombstone does not support nesting of Subroutines which means that only the hole cycle points will be grouped, and not the complete cycles.

Support for Move Angular

Parts programmed using **Move Angular** are now accepted by Tombstone. These will be reprocessed and **Move Angular** items replaced by Indexes which are collision free and allow the user some control over Datums.

Whenever Tombstone finds a part that contains a **Move Angular**, it will replace it by an Index. The Datum generated by that Index will follow the setting for **Absolute** or **Incremental** and be appended to the bottom of the list of Datums (on NC output). The Datums will also maintain their original positioning (as programmed in the original part) to minimise disruption.

A feedback message will also be displayed issued to inform the user that they should review the output and positioning:

'Tombstone - Move Angular has been replaced by Index - please review Datum positioning and output'

Rotated Target positions

Tombstone now allows parts to be inserted at rotated target locations allowing the user to arrange things differently or for when it is a requirement of the Tombstone fixture.

This means a part can be rotated and the engine will ensure that datums align properly with the machine axis for output.

The pre-requisite is that the seed file is created with these target positions; it is the responsibility of the user to define the number of locations and their associated orientations.

When assembling the parts, the Parent part can be inserted at any location and the children can be inserted at target locations that have the same orientation in terms of machine-axis. Should the user wish to insert the same part, but in a different orientation, it must be inserted again, as a new Parent, and the locations must be chosen again.

If a Parent containing Children is changed to a new target with a different orientation, the Children will reset the target and the user will need to select again from a newly filtered list with locations that match the Parent.

Note: Machining / indexing will still fail if the machine cannot physically reach the orientation.

Parts coming from different Machines / Post processors

It is common for the user to program the inserted parts using generic post processors because, at this stage, the type of machine on which the final Tombstone will be executed may not have been defined.

Tombstone now fully supports the insertion of parts that were programmed using a different post processor provided that there is basic compatibility. A warning message will be displayed indicating that the inserted part uses a different configuration but is compatible.

This should provide greater flexibility for large shop floors, with multiple machines, allowing the user to quickly recreate an assembly using the correct Seed / Machine.

Licensing

Tombstone has a dedicated licensing module which controls the availability of the Tombstone browser. The licensing also controls the Tombstone functions on parts with previously created tombstone toolpaths. If an existing Tombstone part is loaded into a session for which it is not licensed, regeneration and post processing will be disabled, with a feedback message being displayed.

Wire Enhancements

Warning if the Offset Side is not as expected for the feature being machined

In this release, we have fixed an issue in Edgecam where the wrong information could sometimes be sent to the Wire engine relating to whether the input geometry was a Boss or a Pocket.

In previous releases, users may have specifically set the Offset Side to Left or Right to overcome this limitation and set the offset on the required side; inside on a pocket and outside on a boss. However, the Offset Side could switch if these parts where regenerated in this release and, therefore, we now check that the Offset Side is as expected for the feature being machined and issue a warning when it is not:

'Offset side is not as expected for selected input geometry - please check that the offset side is set correctly'

Note: Using Automatic Offset Side is recommended and safe.

New Wire Technologies

For this release, new Wire Technologies have been added for the following machine models:

Agie

- AGIE HSS (AA15DE_1 Type)

Fanuc

- FANUC0IE
- FANUC1IE

Makino

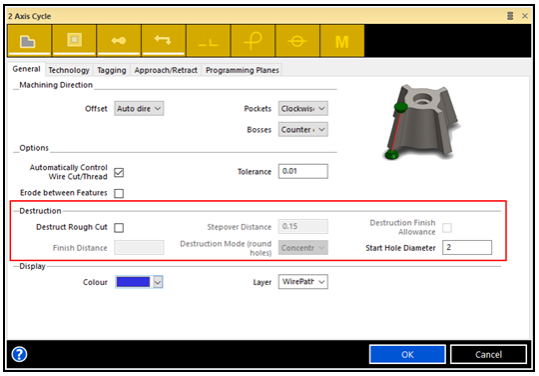
- DUO43 DUO64 (V1) METRIC
- DUO43 DUO64 (V2) METRIC
- DUO43R DUO64R (V1) METRIC
- DUO43R DUO64R (V2) METRIC
- SP43[JP] SP64[JP] (V5) METRIC
- SP43[US] SP64[US] (V5.1) INCH
- SP43R SP64R (V3) METRIC
- U1310 (V3) METRIC
- U1310 (V4) METRIC
- U3 U6 (V1) METRIC
- U3 U6 (V10.1) METRIC
- U3 U6 (V11) METRIC
- U3 U6 (V11.1) METRIC
- U3 U6 (V9.2) METRIC

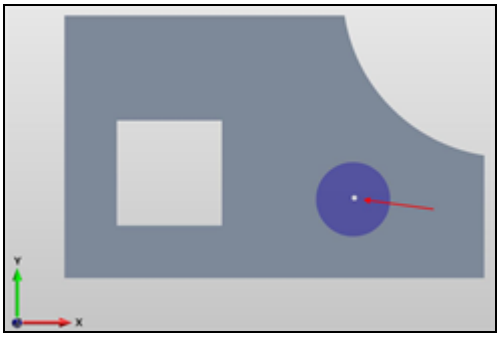
Mitsubishi

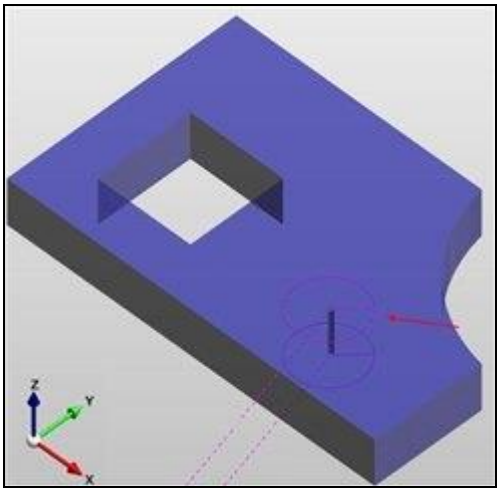
- BRD-B13W032-A13,FA30 (V2.0)
- BRD-B13W062-A8,FA-VS (V10.0)
- BRD-B13W111-A0,NA1200 (V1.0)
- BRD-B13W151-A5,MV2400S_ADVANCE3 (V3.0)
- BRD-B13W158-A3,MV1200S_D-CUBES (V1.3)
- BRD-B13W159-A3,MV1200R_D-CUBES (V1.3)
- BRD-B13W159-A4,MV1200R_D-CUBES (V1.4)
- BRD-B13W160-A3,MV2400S_D-CUBES (V1.3)
- BRD-B13W161-A2,MV2400R_D-CUBES (V1.2)
- BRD-B13W161-A3,MV2400R_D-CUBES (V1.3)
- BRD-B13W161-A4,MV2400R_D-CUBES (V1.4)

Simulator Enhancements

Start Holes in Simulator for Wire cycles







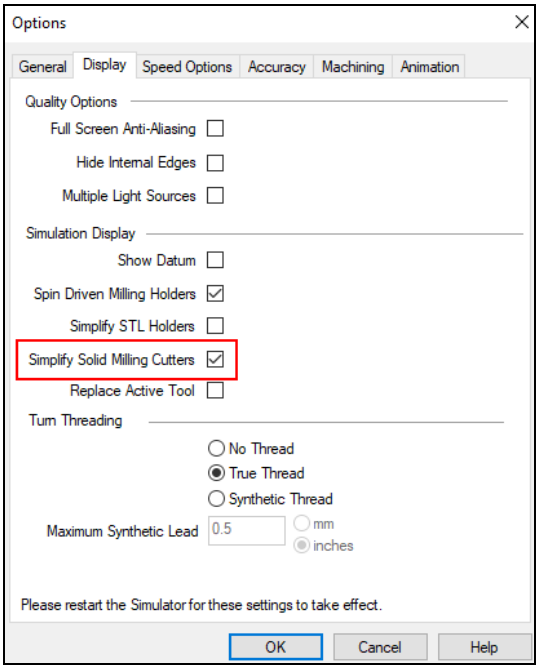
Simulator has now been enhanced to show the Start Holes configured in the Wire cycle.

The Start Hole is where the wire cycle starts. Even though it is usually circular, there are cases when the cycle has used a rectangle with an offset; in these cases, the Start Hole is, in fact, a rectangle and Simulator has also been adapted to show that.

Notes:

- The information passed to the Simulator is currently being used to cut any Stock found in a given coordinate.
- The Start Holes do not relate to a specific stock which allows multiple stocks found in the sequence to be cut.
- The Start Holes are Sequence specific. If there are multiple machining Sequences in a PPF file, the Start Holes will only appear from one of them at a time.

Simulator option to simplify Solid Mill Cutters



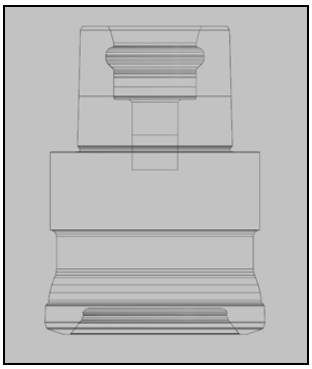
When using solid models for a milling cutter, complicated shapes can cause slow-downs, compromising the overall performance of simulation. This is most significant when there are internal shapes that are, in theory, not relevant to the actual cutting, but still slow down the simulation engine.

To resolve this issue, a **Simplify Solid Milling Cutters** option has been implemented to simplify the process by using a silhouette of the cutter. This removes unnecessary internal shapes, greatly improving simulation speed.

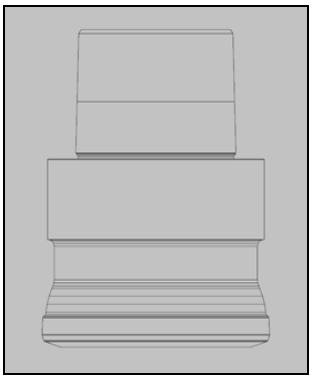
Notes:

- Only affects Milling tools with solid MEG defined as cutter.
- This should be used only when the undercut/internal shape is not relevant to the actual simulation.

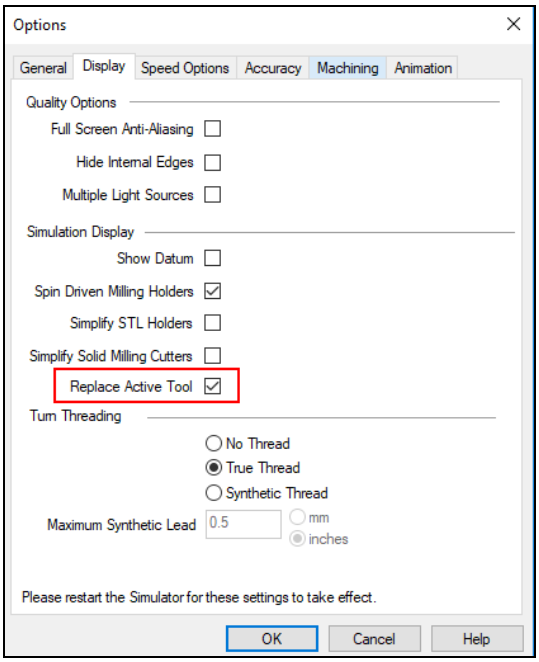
Simplify option OFF - Internal shapes all present



Simplify option ON - Only the outline contour is used to create the cutter



Replace Active Tool in Simulator for Manual Toolchange



For this release, we have added functionality to correctly simulate scenarios in which the machine operator manually changes the tool during the machining process.

The **Replace Active Tool** option enables the Simulator to only show the active tool when there are more tools in the same Turret Position. If more tools are placed in the same Turret Position deliberately when, for example, the positions are controlled by gauge points, this option cannot be used.

Notes:

- This applies to Index and Linear turrets only.
- There are no effects in Edgecam or in the NC Code; the user remains responsible for stopping and replacing the tool.
- Warnings are already displayed in Edgecam, when starting the Simulator, if tools occupy the same position.

Code Generator Enhancements

System Variable for Euler angles

One system variable (EULER1, EULER2, EULER3) has been added to the following Code Generator macros:

- Milling Macro Reference (MACRO 1 - MOVE RAPID).
- Milling Macro Reference (MACRO 2 - MOVE FEED).
- Milling Macro Reference (MACRO 53 - CLW).
- Milling Macro Reference (MACRO 54 - CCLW).
- Turning Macro Reference (MACRO 1 - MOVE RAPID).
- Turning Macro Reference (MACRO 2 - MOVE FEED).
- Turning Macro Reference (MACRO 53 - CLW).
- Turning Macro Reference (MACRO 54 - CCLW).
- C Axis Macro Reference (MACRO 53 - CLW).
- C Axis Macro Reference (MACRO 54 - CCLW).

System Variable for TOOLTYPE

Additional values (23 - Additive Powder Deposition, 24 - Additive Wire Deposition, 25 - Additive Metallization and 26 - Additive) have been added to the TOOLTYPE system variable in the following Code Generator macros:

- Milling Macro Reference (MACRO 15 - TOOLCHANGE).
- C Axis Macro Reference (MACRO 165 - TOOLCHANGE).

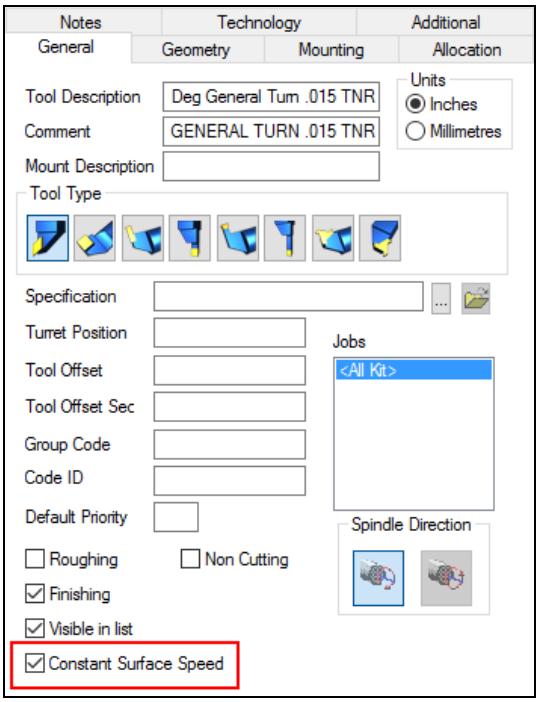
System Variables for Shank Length and Holder Z Offset

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

- Milling Macro Reference (MACRO 15 - TOOLCHANGE).
- C Axis Macro Reference (MACRO 165 - TOOLCHANGE).

ToolStore Enhancements

Edgecam ToolStore - Constant Surface Speed option added for Turning Tools



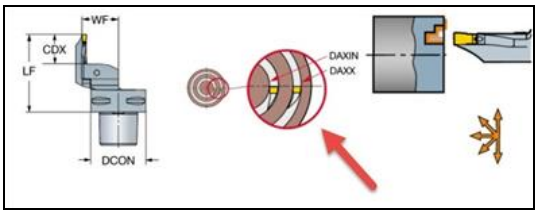
When creating or editing Turning Tool Types in the ToolStore, it is now possible to set Constant Surface Speed (CSS).

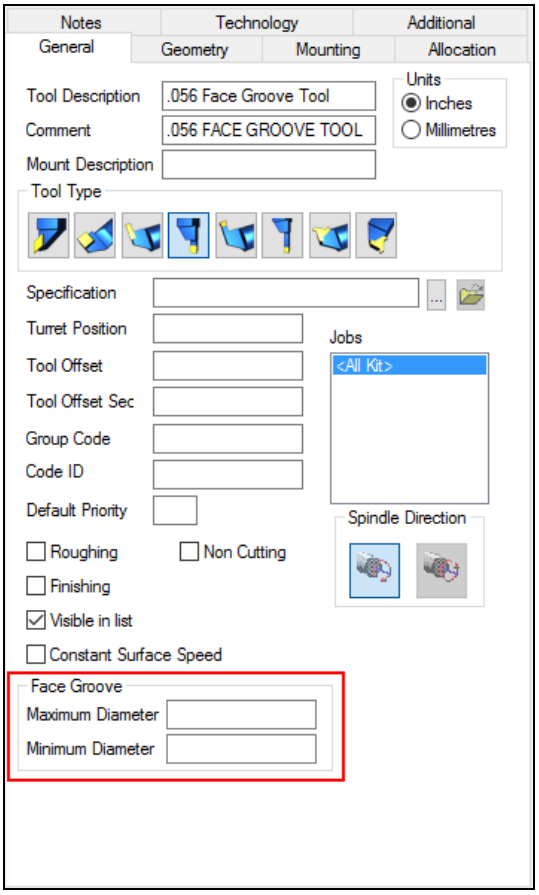
When the tool is selected in Edgecam, the **CSS** checkbox on the resulting Edgecam Toolchange will reflect the ToolStore **Constant Surface Speed** setting.

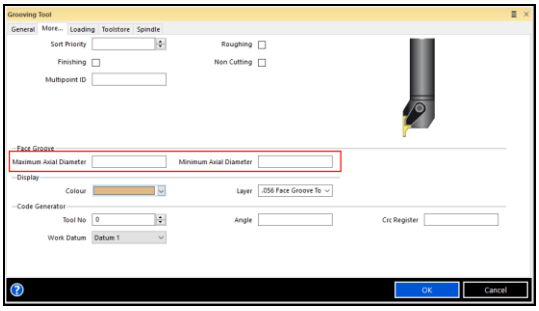
This removes the need to edit the tool in Edgecam each time to set **CSS**.

Note: Not available for Thread tools.

Edgecam ToolStore - Minimum and Maximum Diameter Range modifiers added for Face Groove Tools







Minimum Diameter and **Maximum Diameter** range modifiers have been added for Face Groove Tools; they do not change the geometry of the tool.

Tooling suppliers give the supported diameter range for a tool as shown.

These modifiers are available in ToolStore and Edgecam and can be used when searching the database for a tool in Strategy Manager.

Important Licensing Changes

Sentinel RMS Upgrade to Version 9.2.1

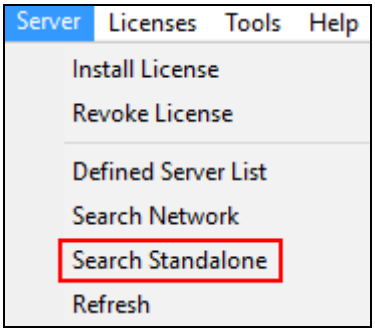
The Sentinel RMS software that our CLS licensing is based on has been upgraded to version 9.2.1.

Customers who have a Network license, must ensure that their **Sentinel RMS License Manager** is updated to version 9.2.1 or higher. After installing Edgecam 2018 R2 and attempting to use network licenses, if the Sentinel RMS License Manager is earlier than version 9.2.1, a warning about the upgrade is displayed.

To install the 9.2.1 Sentinel RMS License Manager, run **setup.exe** from the **\Sentinel RMS Licensing\License Manager** Installation folder of your installation media. Existing network license files are compatible with the new License Server and only require updating to run Edgecam 2018 R2 if the maintenance expiry date is currently March 2018 or earlier.

New 'Search Standalone' option

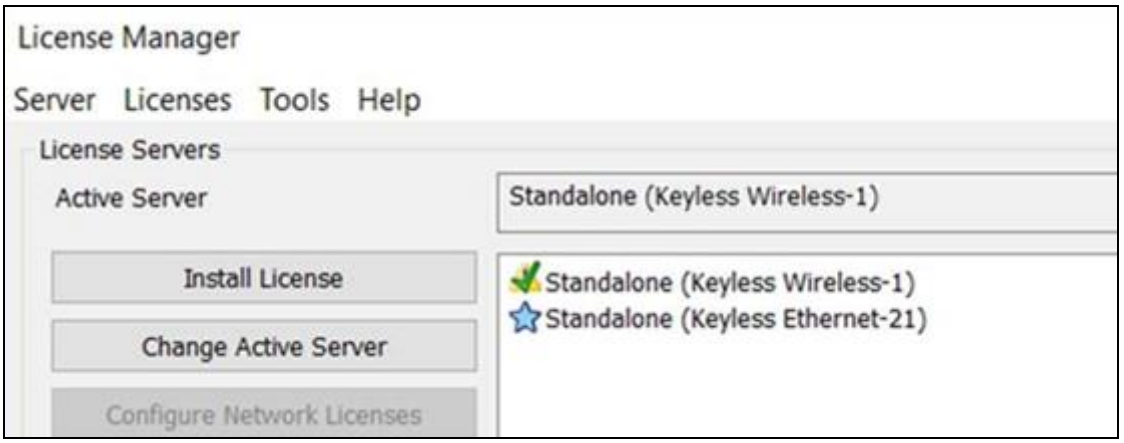
We have introduced the new **Search Standalone** option in the License Manager Server menu to update the list of options in License Manager, for example, when a key has been added to the system. License Wizard will also list all of the Standalone options.



Note that a search will show all keyless options and any valid keys that are attached to the PC. Once the choice has been made and the licenses have been installed, on a refresh or restart of License Manager, only the active option will be displayed.

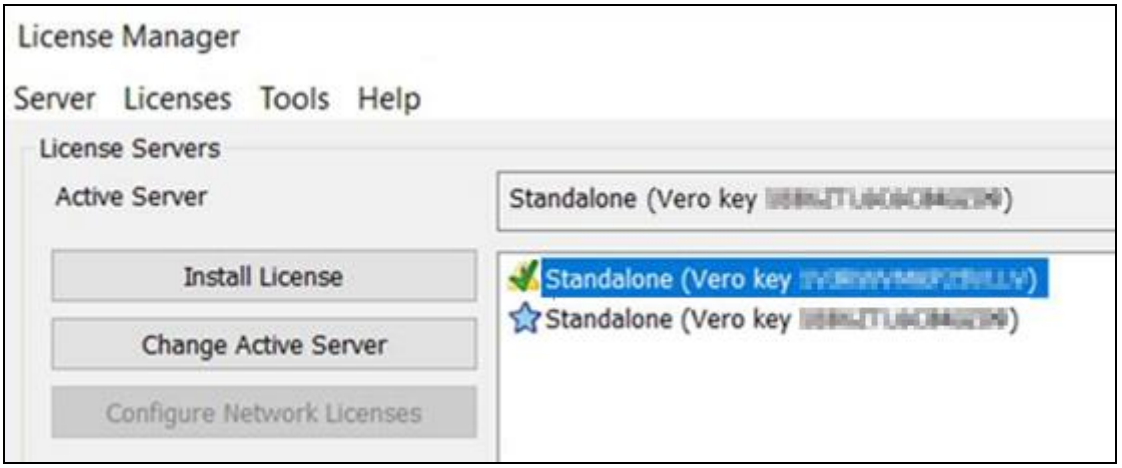
Locking Standalone licenses to Wireless and Ethernet options

To make the management of standalone licenses on laptop computers easier, we now display options for all network adaptors that can be locked to in License Wizard and License Manager, these are labelled Wireless and Ethernet. It is recommended that the Wireless interface option is selected to ensure continued use of the licenses when the laptop is removed from Ethernet cables and docking stations.



Support for multiple Vero Computer ID Keys

CLS now supports the use of multiple Vero Computer ID keys to enable multiple Vero brands to be locked to them on the same PC.



Other improvements in CLS 2018 R2

- When a standalone or network server option is made active but a license for it is not yet installed, the yellow warning triangle is displayed behind the green tick ().
- The time that it takes for the license profile dialog to display and populate the list of profiles has been improved.
- When a network server is made active, it is automatically added to the Defined server list.

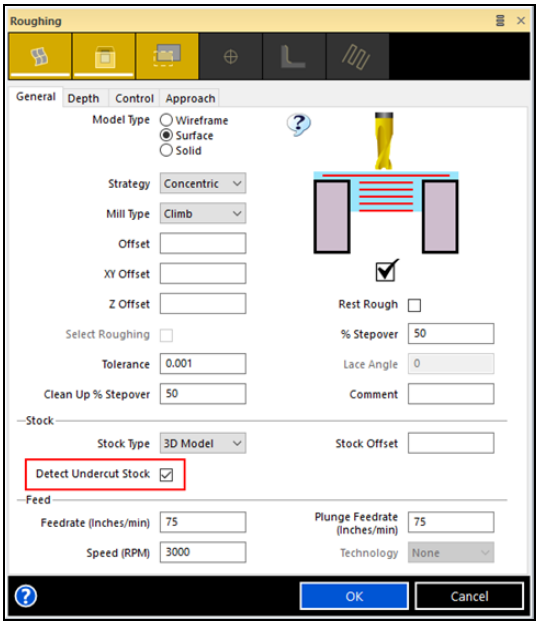
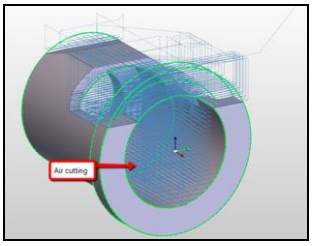
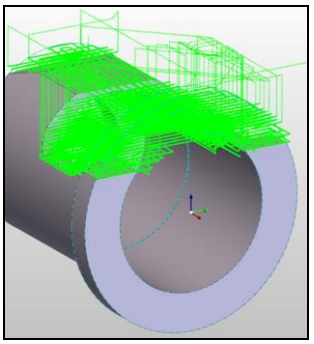
Maintenance Database Report

For a full list of maintenance items resolved in Edgecam 2018 R2, please refer to the **Maintenance Database Report**.

New Features in Version 2018 R1

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](#).

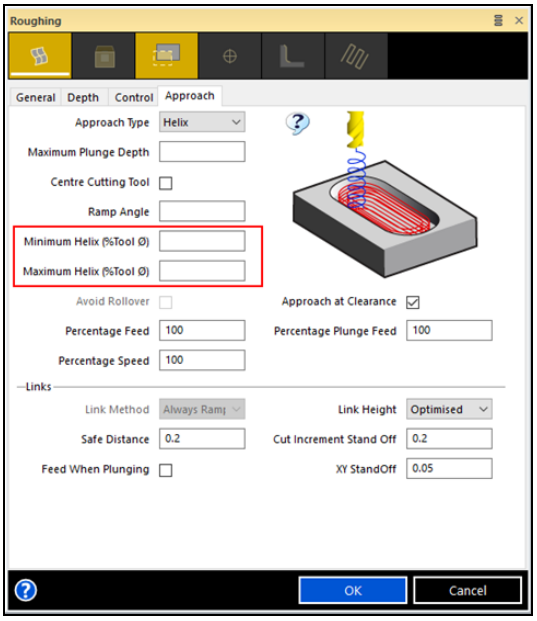
Manufacture Enhancements	
Roughing Cycle - Detect Undercut Stock added to General tab A new **Detect Undercut Stock** field has been added to the General tab of the Roughing cycle. This allows you to analyse the current stock and avoid areas that have no stock: • This option should help to avoid air cutting and reduce the machining time. For this example, the machining time went from 2h 46min to 1h 16min. • The option is an extra calculation step and could add time to the cycle generation time which would not be beneficial on complex parts with no undercut. **Note:** Available on Standard Milling and Standard Production licenses.	
Option not set:	
Option set:	

Roughing Cycle - Waveform - Ability to set Helix Diameter on Approach

For this release, it is possible to define the Helix Diameter on Approach with **Maximum Helix (%Tool ∅)** and **Minimum Helix (%Tool ∅)** when using Waveform strategy for Roughing. Previously, the diameter was automatically set based on the size of the pocket, diameter of tool and whether or not it was centre cutting.

This can be used when the tool allows a specific range of diameters for the helix or when, for a specific application, a fixed diameter must be set:

- The **Maximum Helix (%Tool ∅)** should be used unless limited by the pocket size.
- Equal values for both modifiers will define a fixed diameter for helical approaches.



Pass Feature Boundary to Cycles

When machining a group of features with **Use Current Stock** selected, the cycle also creates a toolpath outside of the part as it tries to machine all existing stock because the limits of the feature are not defined. This can be time consuming and is usually the result when strategies are used.

For this release, we have added an additional **Pass Boundary To Cycles** property to a group of features which will prevent this unnecessary toolpath from being created.

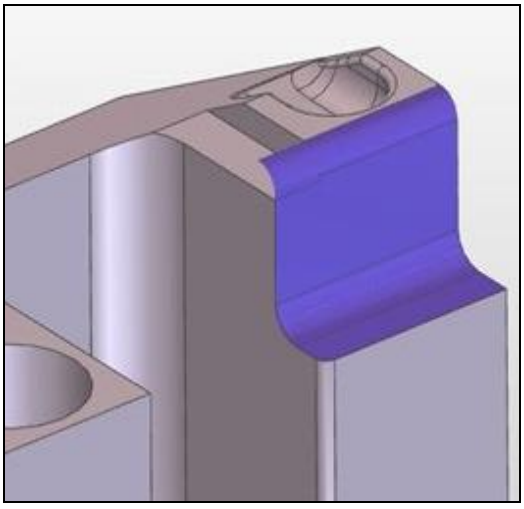
Feature Properties

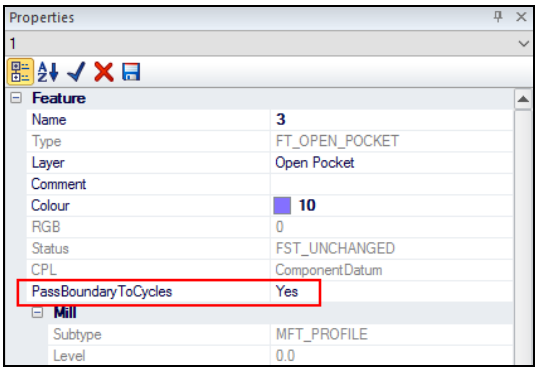
Open Pocket, Open Mill and Flat Face features now have an additional **Pass Boundary To Cycles** property in the Feature section which defaults to **No**.

Cycle Behaviour

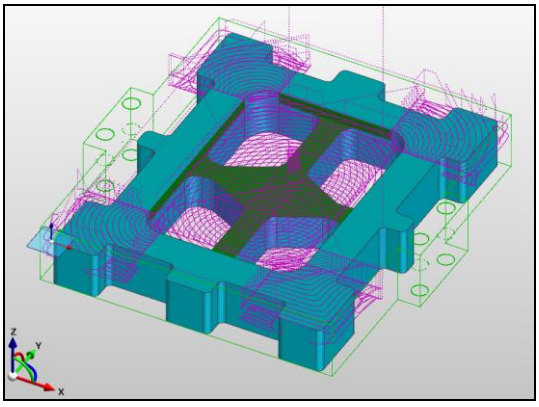
If the **Pass Boundary To Cycles** feature property is set to **Yes**, the Roughing and Plunge Roughing cycle will automatically use the boundary created as shown in the illustration.

Note: The boundaries are generated by extracting an outer silhouette from the bottom face of the feature including any radius that might be connected to the wall.

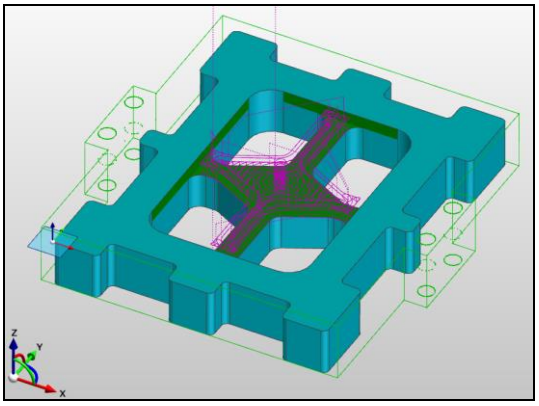




Pass Boundary To Cycles = No



Pass Boundary To Cycles = Yes

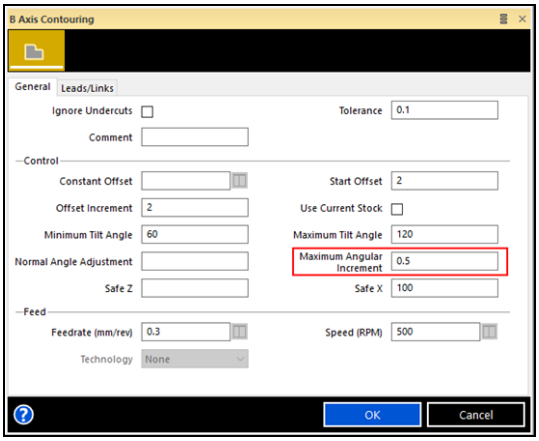


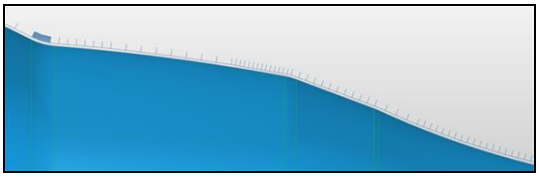
B Axis Contouring cycle - Maximum Angular Increment

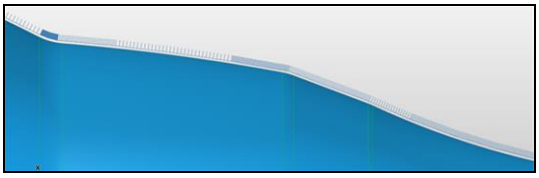
A new **Maximum Angular Increment** modifier has been added to the General tab of the cycle dialog:

- Used to control the maximum angular increment when contouring.
- Results in more or less control points in the toolpath and NC code.
- May be used to help improve the surface finish, especially on machines which do not morph the B axis between the control points.

Checking **Display** in the Normals section of the Preferences dialog - Toolpaths tab shows the effect of this setting.







Rough Turn - Collision Check Chucks

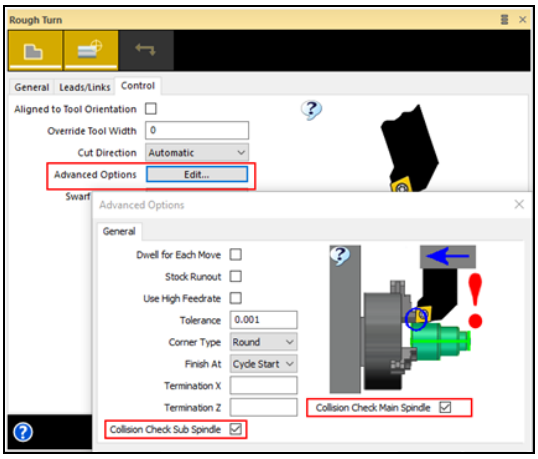
The Rough Turn cycle has been enhanced to collision check the nominated chuck(s).

It uses the **Fixture Offset** in Update Fixtures (General tab) to determine the safety margin.

This is useful when features extend into the chuck and you want to machine them automatically. An additional benefit is that the tool will now cut down the insert side angle rather than just plunge down into material.

Two options have been added to the Advanced Options on the Control tab:

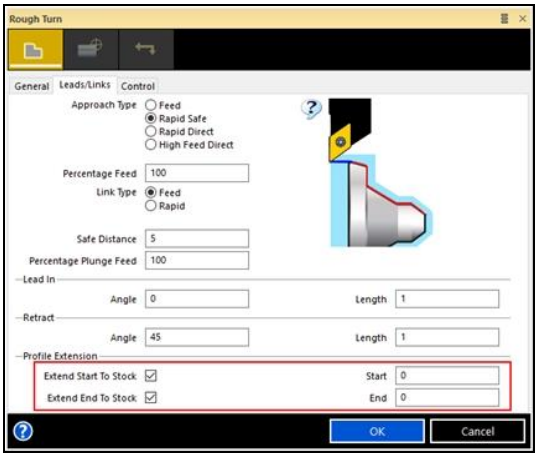
- **Collision Check Main Spindle** - Prevents the tool colliding with the main spindle chuck or jaws.
- **Collision Check Sub Spindle** - Prevents the tool colliding with the sub spindle chuck or jaws.



Rough Turn - Extend Past Stock Edge

For this release, we have added the ability to extend past the stock edge to the Rough Turn Cycle. This can be useful in situations where the stock model is inaccurate or simply to go beyond the stock edge to break off a burr. It can also be useful to extend the start to add some additional clearance when leading in to each cut.

With **Extend Start To Stock** or **Extend End To Stock** selected, **Start** and **End** extension values can now be entered. This extends the profile start / end vectors out tangentially to the stock edge + any extension. It also extends any toolpath stripes above the profile / stock intersection to the stock edge + any extension.



Profiling - Spring Cuts added to General tab

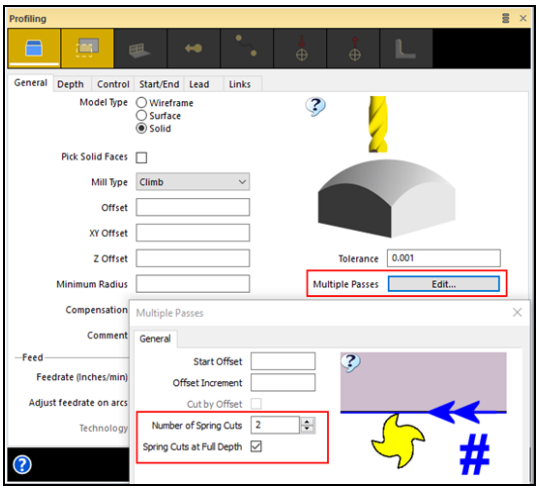
A new **Number of Spring Cuts** field has been added to the General tab of the Profile Milling cycle.

When profiling, tool deflection can result in the part being cut oversize, typically when machining hard materials. It can be necessary to repeat the profiling passes to achieve the correct size and surface finish.

For this reason, we have now made it possible to set the number of spring cuts that are required:

- For multiple passes, the spring cuts are applied to the final cut at each level.
- For multiple cut increments, there is an option to apply the spring cuts only at full depth (**Spring Cuts at Full Depth**).

Note: Available on Standard Milling and Production Licence.



Profiling - Adjust feedrate on arcs added to General tab

A new **Adjust feedrate on arcs** field has been added to the General tab of the Profile Milling cycle. This allows you to adjust the feedrate when going around external and internal arcs.

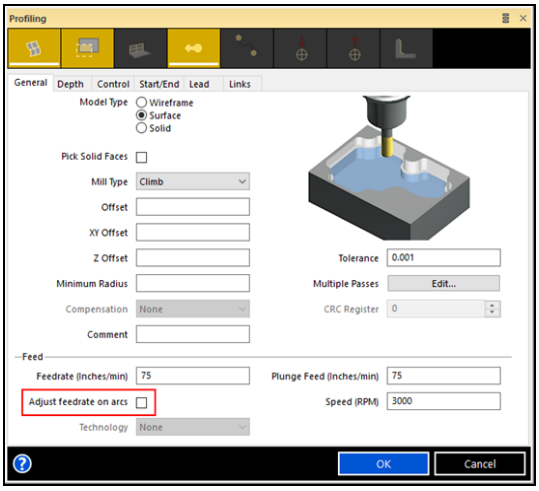
Previously, the feedrate was adjusted in the Code Generator which contained the option. However, using this method meant that any feedrate adjustments were not considered in the Edgecam cycle time.

By making the adjustment in the cycle, Edgecam cycle times now reflect the feedrate adjustments. The following formulas are used to calculate the feedrate adjustment in the cycle:

- External Corner Feed = Linear Feed * (Radius on Part + Toolrad) / Radius on Part
- Internal Corner Feed = Linear Feed * (Radius on Part - Toolrad) / Radius on Part

Note:

1. You should update to the latest Code Generator template to ensure that Code Generator does not duplicate the feedrate adjustments in addition to the cycle.
2. Please ensure that your Maximum High Feedrate is correctly set in your post processor to avoid the feedrate being adjusted beyond the maximum feedrate limit of the machine.



Profiling - Undercut Profiling with Protect Solid is not permitted

In previous versions, the Profiling cycle allowed the **Undercut** and **Protect Solid** options to be set together. However, this combination of options is not supported and causes unreliable results.

For this release, this combination of parameters is not permitted:

- Existing parts that include Profiling cycles with this combination of parameters may change during regeneration in 2018 R1.
- A warning message is displayed if this combination of settings is detected upon regeneration.
- An alternative method should be found for any cycles which are affected, for example, use of boundaries or depth settings.

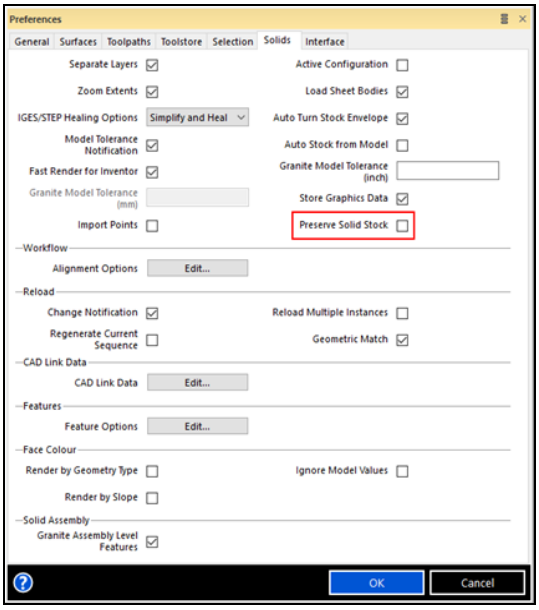
Preserving Features that were found on a Stock Solid

When a solid model is used as stock, you can still find features on it which can be used for cycle creation or for referencing, for example, with 'Move Relative To'.

However, the stock is constantly handled and reprocessed and, potentially, becomes an STL which no longer references the original model. The STL is then discarded (hidden from view) which will then also lose the features.

To overcome this limitation, a new **Preserve Solid Stock** field has been added to the Solids tab which will maintain the solid and its features in, for example, Setup or Spindle Docking.

Note: Existing parts, where a Setup has already been performed, will need to be recreated because the entities in question are already affected.



Speed and Feed Modifiers are now locked when using Technology values from the ToolStore

We have improved the way that the Speed and Feed modifiers work when the Technology values are returned from the ToolStore:

- Previously, it was possible to edit the returned Technology values for Speed and Feed on the cycle dialogs; however, this was unreliable.
- For this release, the Speed and Feed Modifiers are now locked when using Technology values from the ToolStore.
- The returned Speed and Feed values can be modified by selecting Technology **None** and adjusting the values as required.

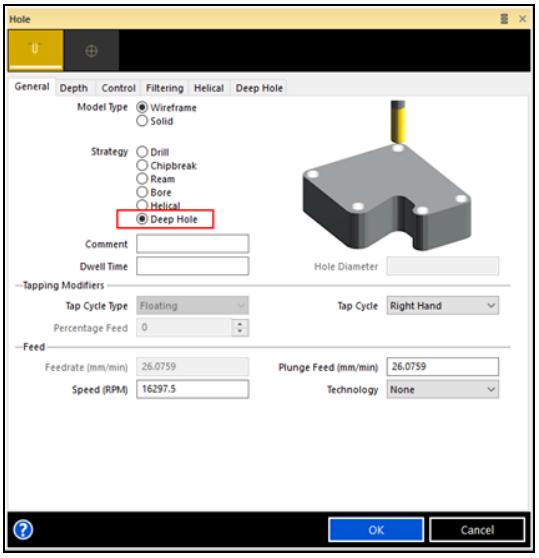
Deep Hole Drilling

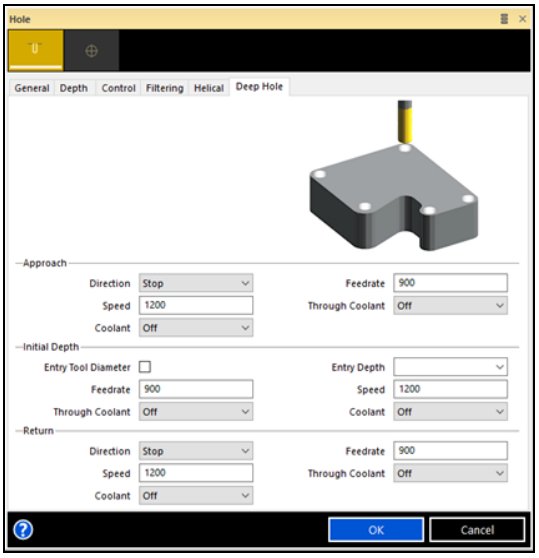
The Hole Cycle has been enhanced with the capability to perform deep hole operations where the user can control key parameters, such as tool direction, speed, feed, coolant and through tool coolant, on different sections of the operations.

This is particularly important where specialised and rather fragile tools are used and the process requires accurate and precise output of such cycle elements.

A new **Deep Hole** strategy is available on the General tab with parameters set on the **Deep Hole** tab.

Note: The license required is Advanced Milling (or above).



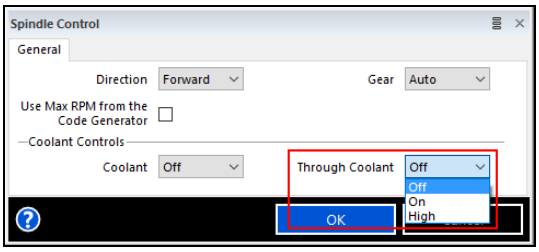


Spindle Control command now controls Through Tool Coolant

The Spindle Control command now controls Through Tool Coolant.

A **Through Coolant** option has been added to the Spindle Control dialog allowing you to specify whether the tool can pass coolant through internal channels and if High pressure is used.

Note: The templates have been updated to support this.



Move To Toolchange / Home on machines with rotary heads

Move To Toolchange / Home has been historically unreliable on machines with rotary axis in the head, when the head was indexed to a plane different from the initial plane.

The main issue was that the moves created (and simulated) did not necessarily agree with what was output. That was obviously not ideal and could lead to a potentially dangerous situation.

To resolve this, we have implemented an option to **Move relative to** the **Machine** or **Datum** orientation on the **Move to Toolchange** dialog:

- Existing commands will default to **Machine** and, therefore, there will not be any toolpath change on existing parts.
- Moving relative to **Datum** means that the movement will take the current Datum orientation and move according to that.

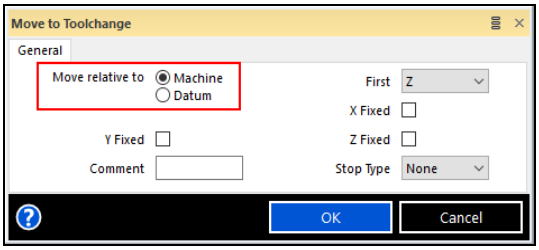
Code Generator

The solution requires changes in post processing to use the intermediate point output which is available on three new system variables: INTERXMOVE, INTERYMOVE and INTERZMOVE.

When these are set, it means that the move is not single-legged. Therefore, to fully benefit from this solution, the post processor will need to be updated to 2018 R1.

Changes will be noticed mainly on head-machines with **Current Datum** output where **Current Datum Coords** is selected because the other modes might be machine-specific.

Note: Templates have been changed so that they extract movement information and output that into the First and Second legs of the move, if they exist. Check your post to ensure that the tokens are placed accordingly, otherwise, the output may not match the created toolpath.



Support for JTOpen file format

Edgecam now loads solids from JTOpen files.

The JTOpen file is a lightweight 3D model format developed by Siemens PLM Software; designed as an open, high-performance, compact, persistent storage format for product data; used for product visualisation, collaboration, and CAD data sharing.

Some of the existing CAD systems that load JT files are: Siemens PLM NX, Siemens PLM I-DEAS, Siemens PLM Solid Edge, Dassault Systemes CATIA, Parametric Technology Creo and Autodesk Inventor.

Notes:

- Existing parts that include Profiling cycles with this combination of parameters may change during regeneration in 2018 R1.
- Edgecam only loads JTOpen files that contain solids.
- PMI data is not supported.
- Versions up to 9.5 are supported.

Handling of Nested features for improved strategy use

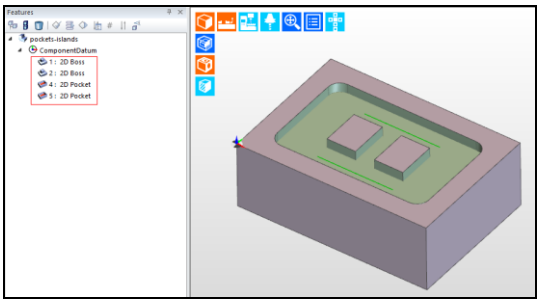
For this release, the Feature Finder has been improved to handle Nested features and reduce duplication. The enhancements should reduce any unnecessary machining that would be created by automatic assignment when using Strategy Manager.

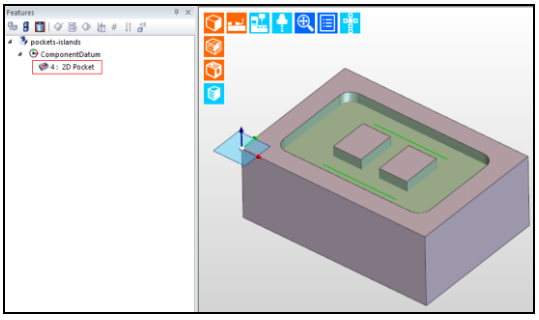
Previously, when finding both Nested and Single Pockets, the Feature Finder would simply maintain all three versions of the feature that were related to the selected geometry resulting in:

- Two internal bosses.
- The internal non-nested pocket.
- The nested pocket, suitable for Strategy Manager.

From this release, if the nested feature can be completely machined, the duplicates will be automatically sent to the bin:

If compatible, only the Nested feature will be maintained and the others will be sent to the bin.





Interface Enhancements

Screen Capture

A new command which allows PCIs to take screenshots has been added for this release.

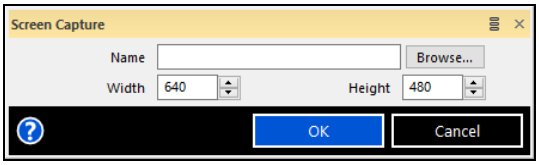
Example PCI-JS code:

```
// Initialising command:- Save JPG
cmd1 = InitCommand(50, 713);
ClearMods(cmd1);
// Setting modifier 'Filename'
SetModifier(cmd1, 56, "c:\\temp
saved.jpg");
// Setting modifier 'Width'
SetModifier(cmd1, 172, "800");
// Setting modifier 'Height'
SetModifier(cmd1, 56, "600");
cmdret = ExecCommand(cmd1, -1);
```

If the Width and Height modifiers are unset, or set to zero, the size of the visible graphics area will be used.

The Screen Capture command can be accessed within Edgecam using one of the following methods:

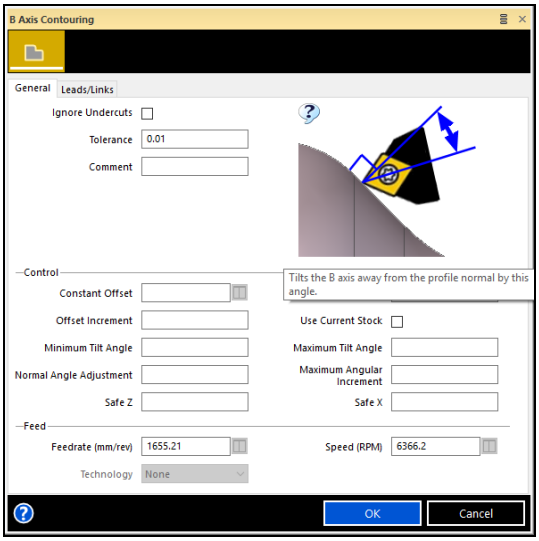
- Type 'Screen Capture' in the Quick Search box in the toolbar.
- Add 'Screen Capture' to the Workflow Ribbon Toolbar.

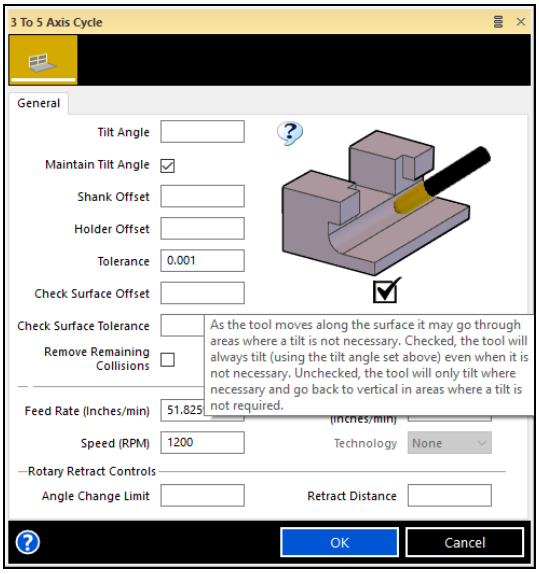


Images on Dialogs

As part of our continuing effort to improve the user experience, images and help tooltips have been added to the following machining cycles:

- B Axis Contouring.
- 3 to 5 Axis.

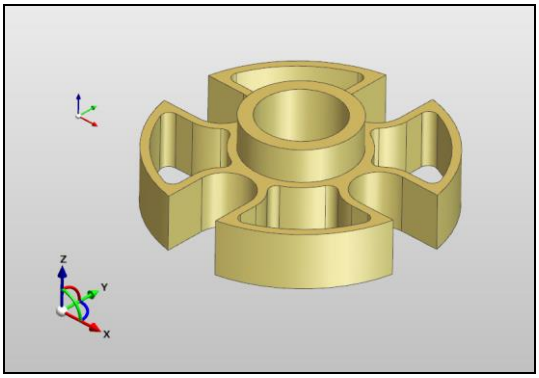


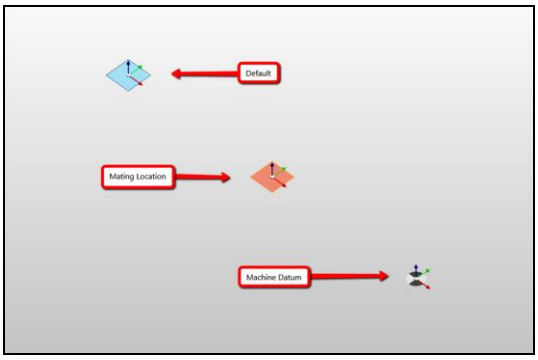


New OpenGL Datum

In order to improve visibility and consistency, we have introduced a new OpenGL Datum for Edgecam. The new datum is bigger and has the same arrow style as the compass (drag-and-drop datum).

We have added different plane indicators to easily identify when working with Default, Mating Location or Machine Datum type of workplane.





PDI API Enhancements

This release includes extensions to the .NET Plugin API which allow plugin developers to:

- Create billboarded information panels, 3D arrows, and labels. These graphical elements can be used to display information to users in a convenient fashion.
- Show and hide reports. In the main window, the context menu, accessed by right clicking, includes an option to show and hide the reports. The option is only available if reports have been created.

Formulas in Dialogs

Following the work to implement Formulas in Dialogs for the 2017 R2 release, a number of enhancements have been implemented for this release which will help users to access the formulas and identify when they have been used:

Auto Complete

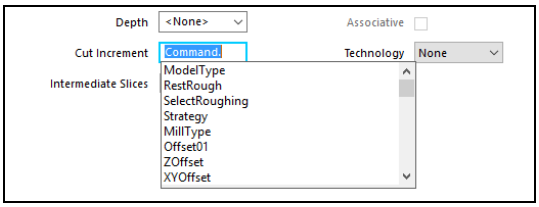
When you type a valid object into the modifier and then add a dot, a list of valid formulas for the given dialog will be displayed, for example:

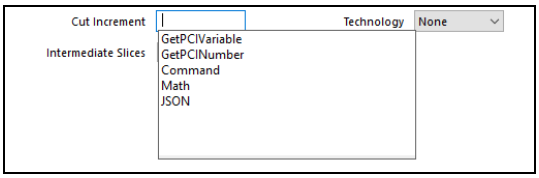
Command.
Math.
GetPCIVariable.

Alternatively, you can select the field in which the formula will be used and press Ctrl + Space to display a list of all possible objects. Use the keyboard arrows to navigate to the required formula and press TAB to transfer the formula to the field.

Highlighting variables

Modifiers containing variables are highlighted in bold with a cyan boundary.





Simulator Enhancements

Spinning effect on Holders during simulation

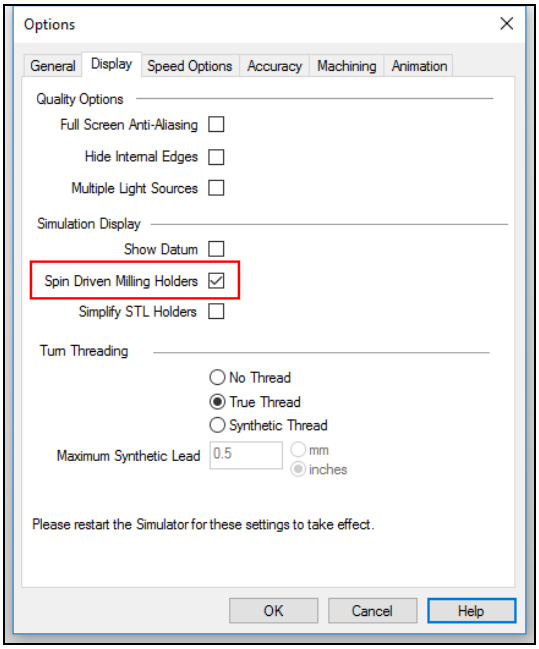
Holders of driven tools can be spun in the simulator to represent the behaviour of the actual machine tool. This is controlled by the **Spin Driven Milling Holders** option.

Previously, when **Spin Driven Milling Holders** was selected, driven tools would always spin the holder, irrespective of the tool direction.

For this release, when the option is selected, the Simulator will only spin the holder if the **Direction**, set on the Spindle tab of the Edgecam cutter dialog, is not **Stop**.

This is particularly relevant for Probing and Additive tools.

Note: This does not affect the spinning of the cutter. A driven cutter will continue to spin as this is essential for cutting the stock.



Code Wizard Enhancements

Euler angles for Multi-plane Machining

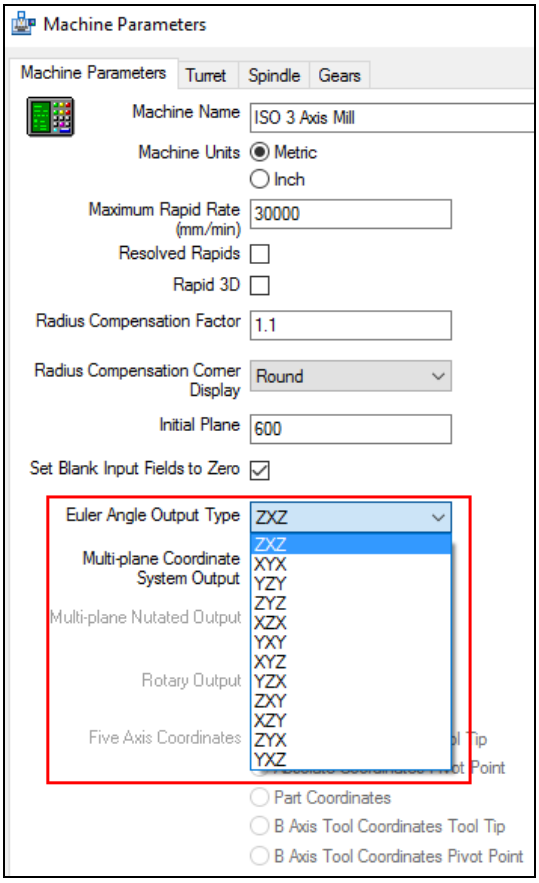
Euler angles can define a coordinate system by a set of three composed elemental rotations. Usual definitions include:

- Proper Euler angles which can be Z-X-Z, X-Y-X, Y-Z-Y, Z-Y-Z, X-Z-X, Y-X-Y.
- Tait–Bryan angles which can be X-Y-Z, Y-Z-X, Z-X-Y, X-Z-Y, Z-Y-X, Y-X-Z.

These angles can be used on index and for describing the orientation of an angled-head. Complex machines and Robots also typically require Euler angles.

For this version, we are extending the capabilities of Edgecam to output these angles; only Z-X-Z definition was available previously. Now the user will be able to configure, in the post, what set of composed rotations is to be output.

Output continues to be through the same system variables and tokens.



Code Generator Enhancements
System Variable for Euler angles One system variable (EULER1, EULER2, EULER3) has been updated in the following Code Generator macro: • Milling Macro Reference (MACRO 43 - INDEX PALLET). One new system variable (EULER1, EULER2, EULER3) has been added to the following Code Generator macro: • C Axis Macro Reference (MACRO 253 - B AXIS INDEX).

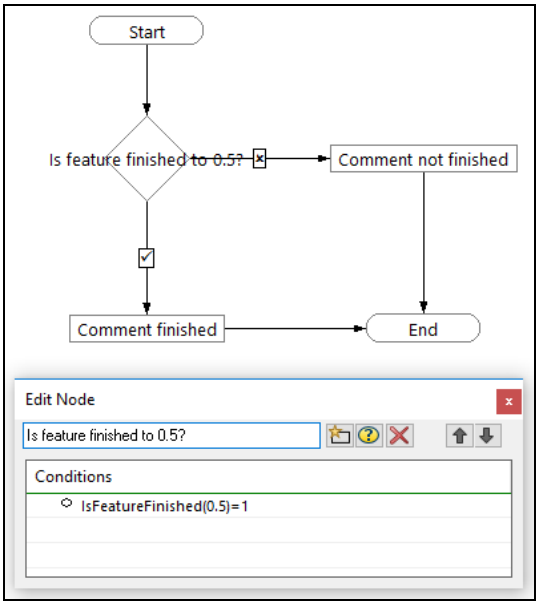
Strategy Manager Enhancements

Strategy Manager - IsFeatureFinished function for Holes

The Strategy Manager IsFeatureFinished function has been enhanced to detect whether holes are finished to a given offset:

- It compares the selected feature against the current stock model and finds the maximum difference between the two.
- Use IsFeatureFinished to determine if the feature is finished to a given offset in order to determine if further machining of the feature is required.
- When setting the offset to check against, you should take into consideration the inaccuracies of the stock model.

Note: Rotary holes are not supported.



Important Licensing Changes

Support for Edgecam legacy Sentinel and HASP keys removed

The Edgecam legacy keys listed below are not supported in Edgecam 2018 R1 and future releases:

- Edge TimeHASP (USB).
- Edgecam MemoHASP.
- Edge NetHASP (USB).
- Edge TimeHASP (PP) (Blue Key).
- Standalone Network (Green Key).
- Network NetSentinel (Yellow/Red/Grey Key).
- Full Customer SuperPro (Blue Key).

If you have one of these keys and have not been contacted by your Vero representative yet, please contact them to arrange an exchange for a Sentinel RMS license. The license type is listed on your delivery note.

Sentinel RMS Upgrade to Version 9.1

The Sentinel RMS software that our CLS licensing is based on has been upgraded it to version 9.1.

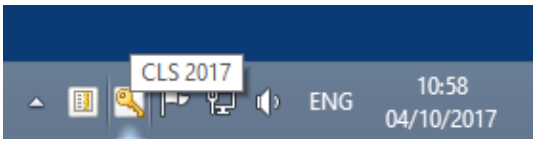
For customers who have an existing Standalone license (keyless or locked to a Computer ID key – see image below) your licenses will automatically be configured for use with Edgecam 2018 R1.

For customers who have a Sentinel RMS Network license, you must ensure that your **Sentinel RMS License Manager** is version 9.1 or higher. After installing Edgecam 2018 R1 and attempting to use network licenses, if the Sentinel RMS License Manager is earlier than version 9.1, the message shown below will be displayed.

To install the new Sentinel RMS License Manager, run **setup.exe** from the **\Sentinel RMS Licensing\License Manager Installation** folder of your installation media. Existing network license files are compatible with the new License Server and only require updating if the maintenance expiry date is September 2017 or earlier.

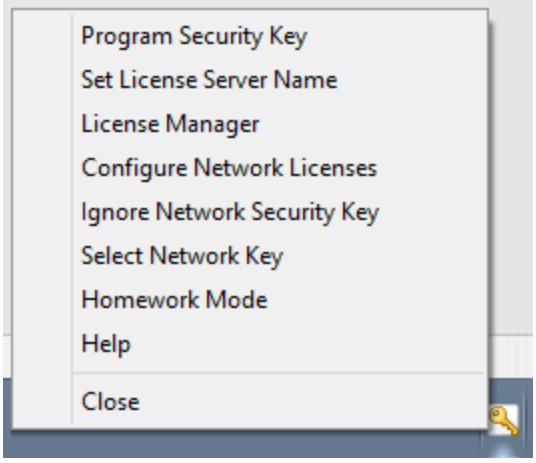
How to access licensing in 2018 R1

Edgecam 2018 R1 introduces a new version of CLS licensing and the CLS icon that was previously in the notification area at the bottom right hand corner of your desktop has been removed. Note that previous installations of CLS for versions up to and including 2017 are not removed and can still be used for licensing the respective versions of the software.



CLS 2017 was the last release to display the CLS icon.

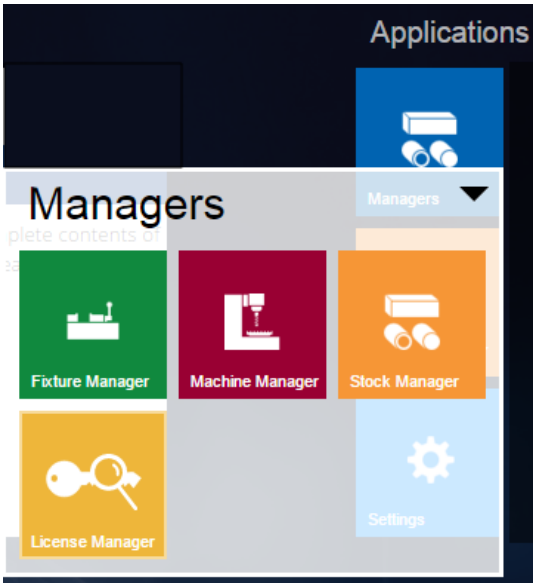
The License Manager, Configure Network Licenses, Homework Mode, Preview and Help menu options that were previously displayed on the CLS menu are all now in the License Manager.



The CLS Menu in previous releases.

Note: The **Program Security Key**, **Set License Server Name** and **Ignore Network Security Key** options are no longer required because they were only relevant to the legacy keys for which we have removed support.

The License Manager application is now accessed from the Edgecam Launcher.



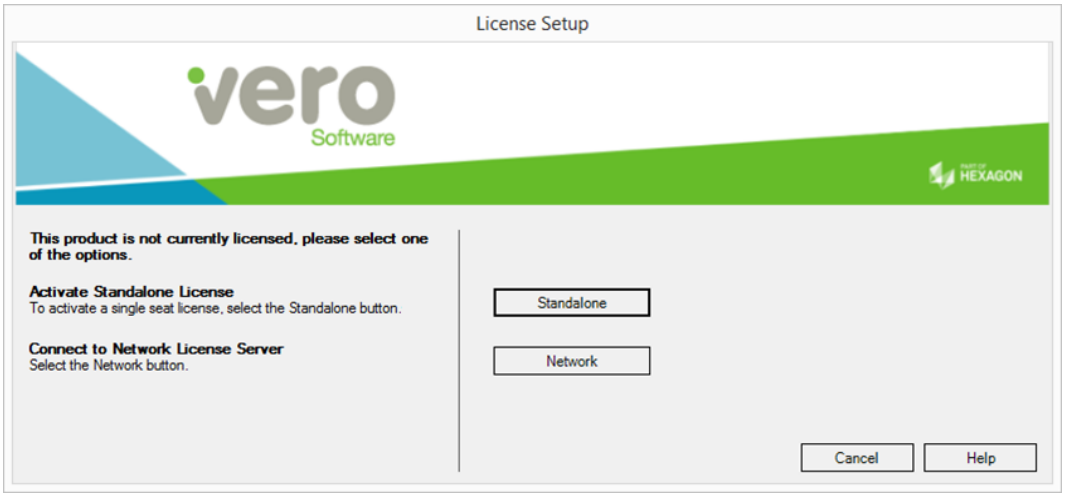
The Configure Network licenses option is accessed from a button on the License Manager dialog.

The Homework Mode and Preview options are accessed from the Licenses menu of the License Manager.

Note: The Preview option is a licensed option that is available on request.

New Licensing Wizard

The process for licensing Edgecam on a system that has not previously been licensed has been made easier with the introduction of the Licensing Wizard. When the Edgecam Launcher or Edgecam shortcut is selected on an unlicensed system, the License Wizard is launched and will guide you through the licensing process.



Users who have exchanged their legacy key license for a Sentinel RMS license will use the License Wizard to configure their new license.