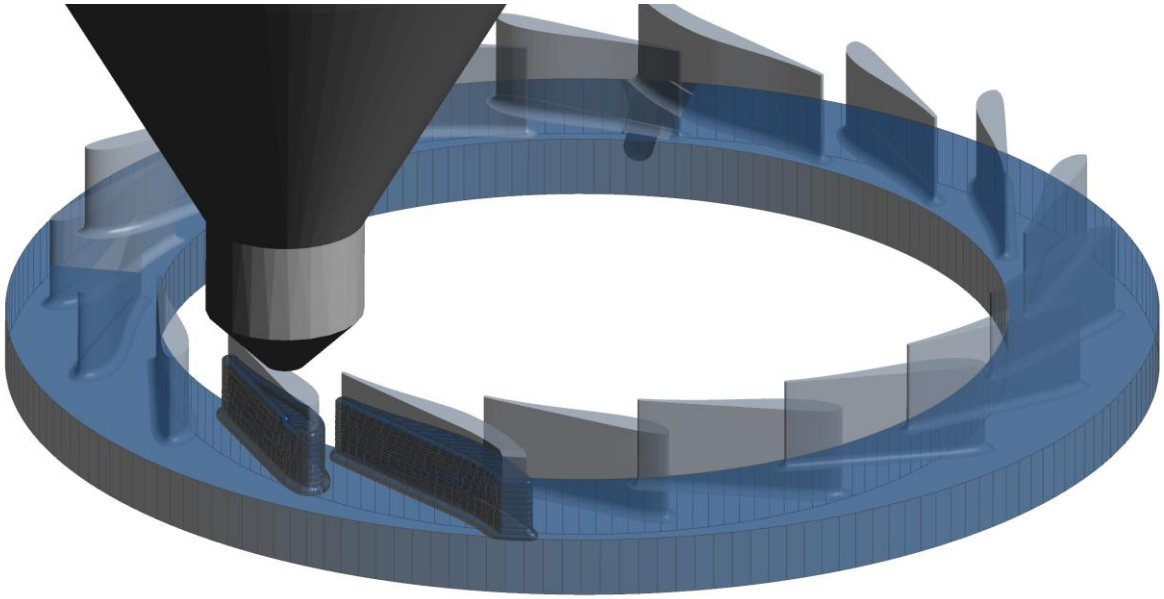
What's New in Edgecam 2019 R1



This document highlights new product features and enhancements in Edgecam 2019 R1.

To run Edgecam and Part Modeler 2019 R1, the maintenance expiry date in the license must be October 2018 or later.

Contents

ToolStore Enhancements

‘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 R2.

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

Important Information

Solid Machinist for Granite removed from the price list

Solid Machinist for Creo is now the option that should be selected for supporting the loading and feature finding on Creo models. Existing Solid Machinist licenses will continue to work as normal.

Additive Manufacturing

Additive capability is now licensed and available for purchase. The Edgecam Additive Machining (ENADD-M) module requires an Advanced or Ultimate licence as prerequisites.

Autodesk Vault

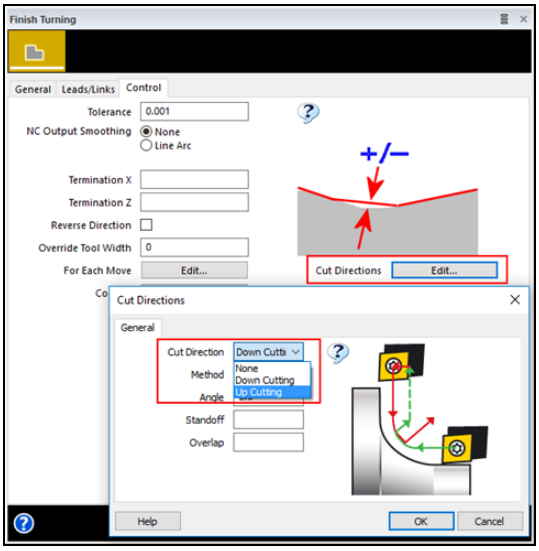
Autodesk Vault has been removed from Edgecam, CLS and SMP.

Part Modeler

The April 2019 release will be the last version of Part Modeler to be supported or sold. We will be recommending that our customers move on to Designer.

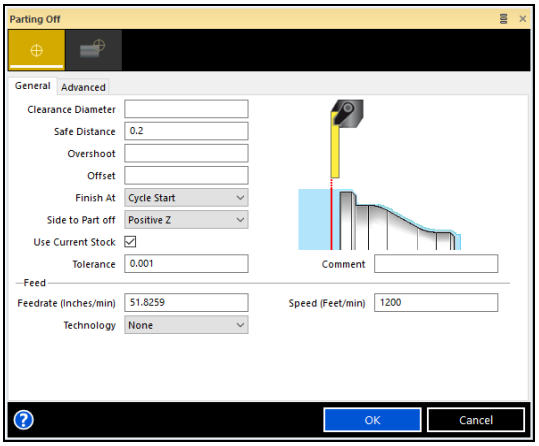
Manufacture Enhancements

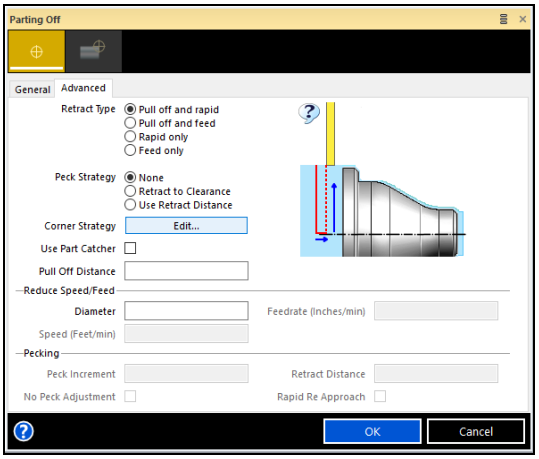
Finish Turning - Up Cutting option



An **Up Cutting** option has been added to the **Cut Direction** option of the Control tab which, when checked, causes the selected turn profile to be split into steep and shallow regions; forcing the Cut direction on the steep regions to be Up Cut, which is the preferred cut direction for some tooling inserts.

New Parting Off Cycle

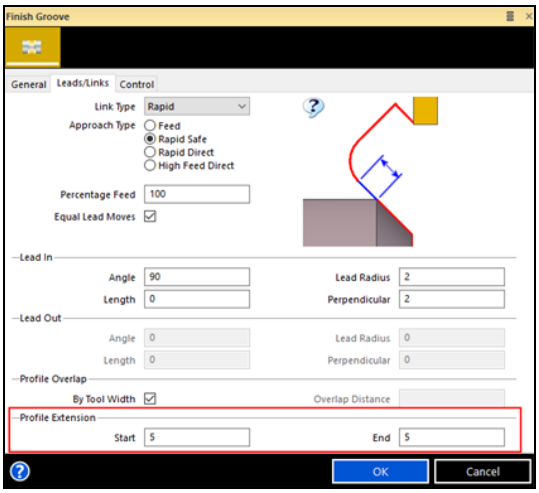


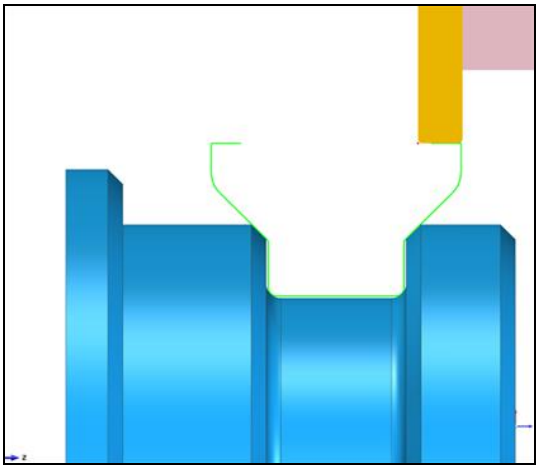


A new Parting Off cycle has been introduced for this release which is used to separate the machined component from the stock:

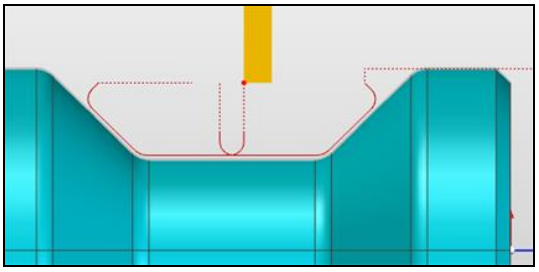
- The cycle will work for both external and internal Parting Off.
- The cycle will work for main and sub spindles.
- The cycle will work for upper or lower turrets.
- For internal Parting Off, the cycle will only work with Current Stock.

Finish Groove - Start and End Extensions





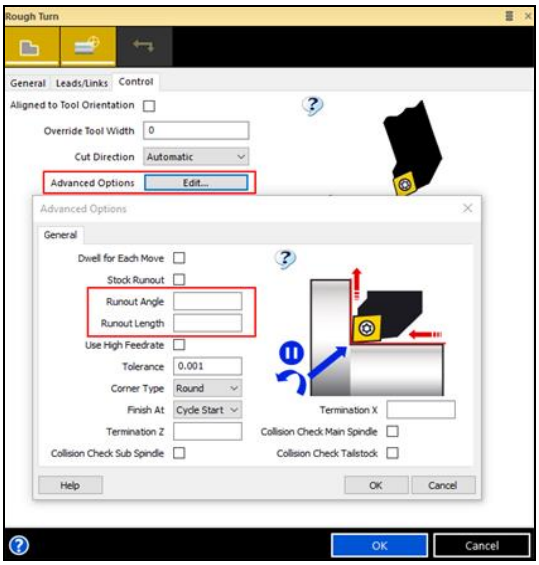
Negative Extensions

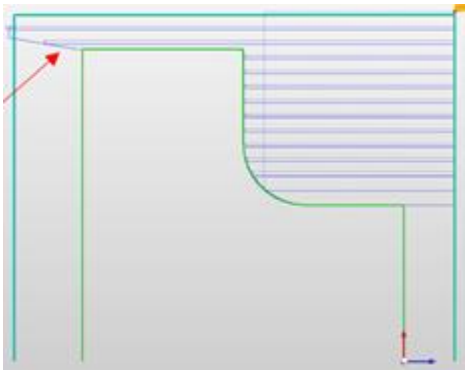


It is now possible to specify **Start** and **End** extension values in the Leads/Links tab of the Finish Groove cycle. Previously, this was only possible by dragging the Start/End points.

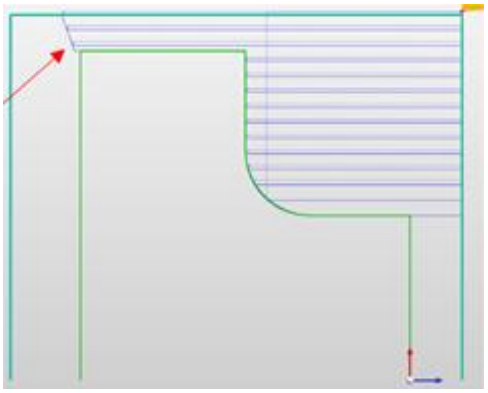
Note: Positive or negative extension values can be applied.

Rough Turn - Added Stock Runout Angle and Length





Runout Angle = 10º



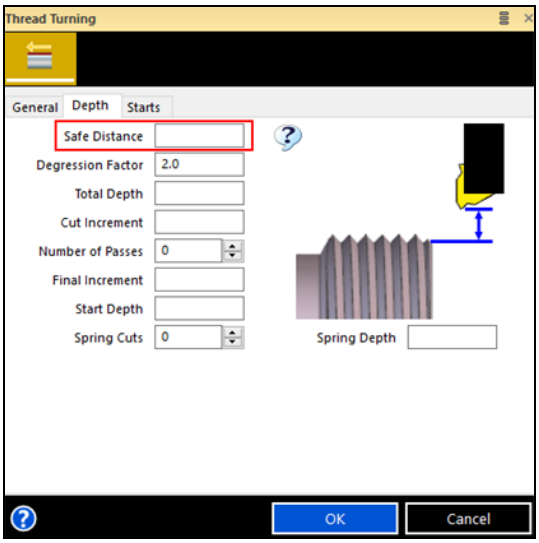
Runout Angle = 70º

The **Stock Runout** option in the Rough Turn cycle was available in previous versions of Edgecam. This parameter allowed each cutting pass to be extended out towards the stock boundary. For this release, two new options have been added:

- The **Runout Angle** will modify the angle of the extension. In the example, an angle of 10º is used in the first image and an angle of 70º is used in the second image.
- The **Runout Length** can be used when the runout needs to be trimmed because it would be too long in some situations.

Note: The **Runout Length** does not have a higher priority than the original Runout element length and will only be applied if bigger than the original element length.

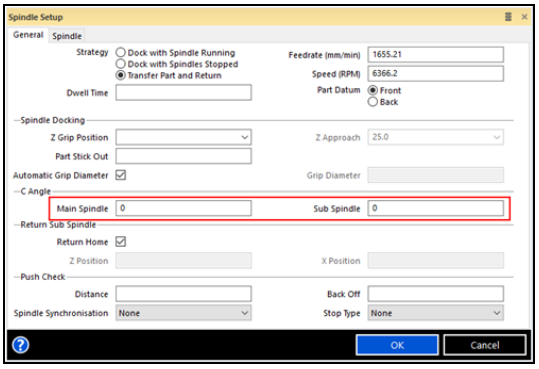
Thread Turning - Added Safe Distance



For this release, a new **Safe Distance** setting has been added to the Depth tab of the Thread Turning cycle.

This value represents the safe distance between passes which is always applied orthogonally to the input profile. In previous versions, this parameter was set from the Miscellaneous menu but it is now available in the dialog.

Controlling spindle C-angles on docking



For this release, we have provided the capability to control the Main and Sub spindle C-angles during a Spindle Setup.

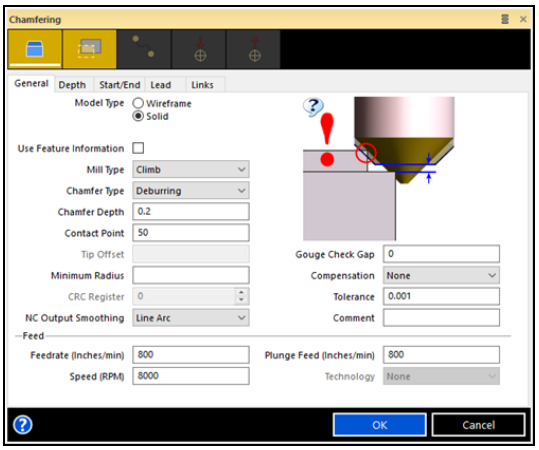
New **Main Spindle** and **Sub Spindle** C-Angle modifiers have been added to the General tab of the **Spindle Setup** dialog which will be available provided the machine has C axis control capabilities.

When performing a transfer, the following events will take place:

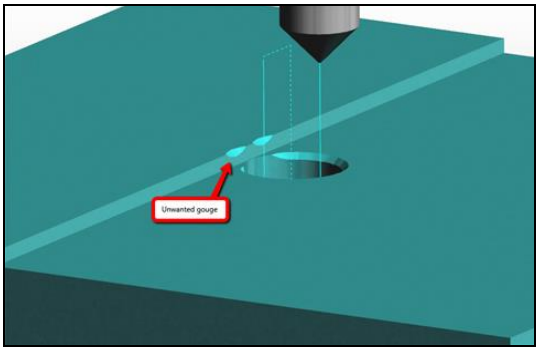
- Donor spindle is selected.
- Donor spindle is rotated, using **Move Angular**, to the required angle.
- Receiving spindle is selected.
- Receiving spindle is rotated, using **Move Angular**, to the required angle.
- Main spindle is selected.
- Sub spindle moves to approach and then final grip position.
- Component is handed over to the receiving spindle.
- The component (stock, solid, features) is now rotated by the resultant angle of the two spindles, replicating the physical resultant rotation that took place in the machine.

Note: If the machine has an offset Sub Spindle (in X), controlling the C-angle is not supported.

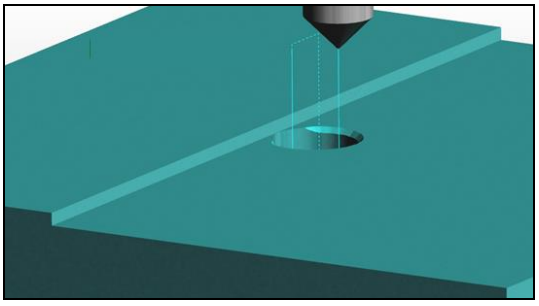
Chamfering - Gouge Check Gap option



Gouge Check Gap set to zero (disabled):



Gouge Check Gap set to default (empty):

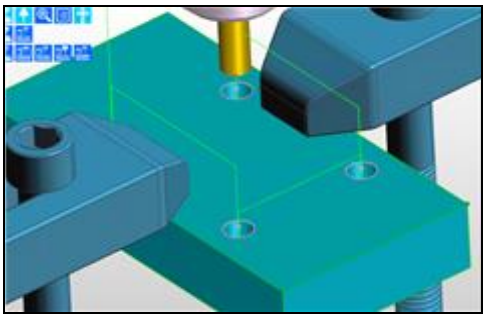


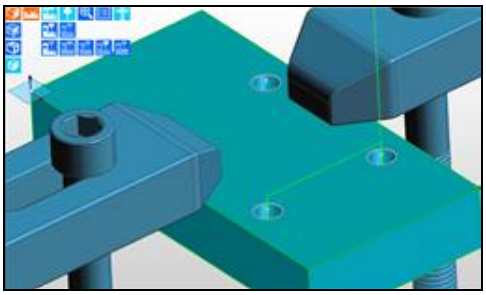
In order to avoid unwanted gouges on edges that were not selected for the cycle, a new **Gouge Check Gap** option has been added to the General tab of the Chamfering cycle. The option is only available when **Chamfer Type** is set to **Deburring**.

This helps the user to control how close to the wall the tool can go. By definition, the **Gouge Check Gap** is the distance above the selected edge where the tool is no longer allowed to gouge the model:

- Leave empty for the default value (half of the **Chamfer Depth**).
- Use zero to disable gouge checking. Existing parts have a default value of zero which disables the gouge checking.

Collision Check for Hole Cycle (obstructed holes)





In the Hole Cycle, the Fixture Collision Check will raise the Level of link moves in order to avoid collisions between the Tool / Holder and the Fixture. In Edgecam 2019 R1, it will also avoid all Holes that are obstructed by a Fixture:

- In Longhand output, the output coordinates will follow the toolpath.
- In Canned output, the coordinates of the missing holes will be removed from the NC file. The previous behaviour will be maintained with regard to the Link moves where one Canned Cycle will be split into multiple Canned Cycles in order to correctly control the Link moves in the NC code.
- A message will be displayed in the Feedback window to warn users that holes have been avoided: '*n* holes have not been machined as the tool would collide with a fixture'.

Note: Users may see differences during Sequence regeneration. Since it follows the configuration of the last **Update Fixtures** instruction, there could be situations where this will need to be adjusted.

Profiling - Behaviour change if using CRC Geometry path when a Taper Tool is active

In 2018 R2, a check was added to the Profiling cycle which disallowed the CRC Geometry path when a Taper Tool was active. Regeneration of existing parts that included this combination would stop and the user would be prompted to select CRC **Compensation** = **Centre Line** or **None**; the safe option.

For this release, this behaviour has been changed to allow existing parts with this combination to regenerate without stopping and write a warning to the feedback window instead which is the same as releases prior to 2018 R2.

We now also allow this combination for a new cycle but issue the following warning:

Extreme care should be taken when using this combination as in some situations the back offset can self intersect and the resultant toolpath can gouge the component and this is not detected in the Simulator.

The Profiling Cycle CRC back offset is based on the major diameter of the tool; some users adjust the major diameter / radius on the machine controller or define the tool in Edgecam with the major diameter set to the contact point diameter. To ensure that these methods continue to work we now allow this combination of Taper Tool and CRC Geometry path with the caveat that extreme care should be taken.

Edgecam Inspect improvements

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

- **Switch from Wilcox Gateway to PCDMIS fit libraries**

  Edgecam Inspect 2019 R1 will start using the PCDMIS fit libraries, which have certified and approved math, and include some new functionality such as the evaluation of Cylindricity and Conicity.

- **Associative Clearance**

  The cycle now offers an option for the Clearance to be associative with the solid model:

  - Check the option for the Clearance level to be incremental from the highest point of the solid and associative to the solid model.
  - Uncheck the option for the Clearance value to be relative to the active Work Datum.

- **Option to Create Layers**

  An option has been added to the Options menu which allows the user to choose between creating the features on predefined layers or at the active layer.

- **Evaluation of Axis Deflection**

  Edgecam Inspect now offers an option to evaluate the Axis Deflection of 3D Cylinders/Cones (more than 1 level) and Plane features.

  The option to calculate and show it in the report file is found under the Advanced tab.

  In addition, more feature properties are shown in the report, such as number of levels and touches per level.

- **Allow Edge feature in rotary faces**

  Edge and Edge Angle features can now be created in rotary faces, for example, between cones and cylinders.

- **Improvements for Distance Constructive features**

  Distance To Point and Distance To Line features can now be created based on Edge and Rectangle features.

  Also, it is now possible to set the Distance Type prior to creating the feature.

- **Manual features**

  In Edgecam Inspect 2019 R1, it is possible to create inspection features by entering manual coordinates.

  It allows the user to create features even when not using a solid model or on a model with incorrect geometry.

  To use it, check the **Manual Input** option (Options menu > Features tab) and then, when creating a feature, a dialog will displayed allowing you to enter the coordinates.

- **Allow Arc feature to be indexable**

  Arc features can now be measured with any stylus orientation.

  The two limitations are that the arc must not exceed 180 degrees and the arc must be processed collision free with the requested tool orientation.

- **Constructed Circle**

  Constructed Circle allows the user to create construct circular features (circle/arc) based on existing single pointed features.

- **Option to select type of Work offset update**

  Work Offset update now has two options:

  - **Type** which defines which table you want to update (work offset or error compensation).
  - **Axes** which allows you to select which axes are to be considered when updating.
- **Hide the Processor section of Options dialog**

  We have removed the Processor section from the Options dialog.

  Demo Mode and **External** are now turned off by default.

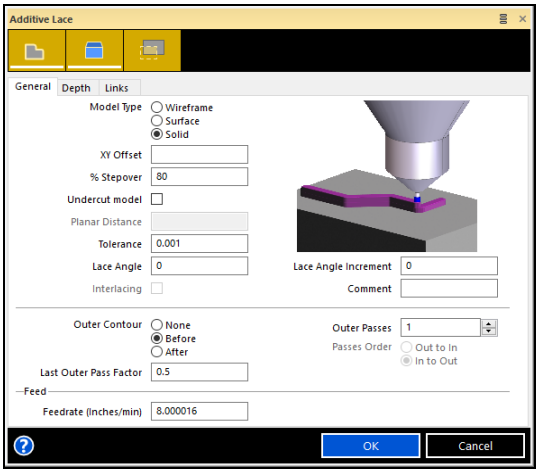
  Setting the **EIShowProcessor** PCI variable to 1 enables this section to be displayed again.
- **Custom Report**

  Edgecam Inspect now supports custom reports.

  This improvement allows the user to develop his own plugin and generate the report file of the measurements customised to their requirements.
- **Support of canned cycles**

  Edgecam Inspect Canned Cycle was developed to allow probing cycles to be implemented.

Additive / Hybrid machining



Edgecam now officially introduces support for additive technology, which, when combined with the existing 'material removal' cycles, forms what the industry calls Hybrid Manufacturing.

The functionality is fully licensed from 2019 R1 and available for purchase. The 'Edgecam Additive Machining (ENADD-M) module requires Advanced or Ultimate as pre-requisites.

The main physical aspect of this is that the Additive Head needs to be vertical and the surface horizontal or near horizontal; there is little adhesion to the wall although undercuts can be constructed gradually:

- In Edgecam, additive cycles can be built using virtually any cycle, or even manual moves. Leads, links and rapid moves are non-additive,
- Advanced cycles, such as Rotary and 5-axis, can also be used, where the same characteristics apply, though with less and more difficult control.
- A dedicated 'filling up' cycle is available, Additive Lace, designed to construct geometry on a layer-by-layer basis, bottom-to-top.

The Additive Lace cycle has the following features:

- **Outer Contour**

  Toolpath buildup can result in discrepancies and runoffs on the external edges where the melt pool is more vulnerable to gravity effects.

  For this reason, an **Outer Pass** is required, either before or after the internal region is filled. This is similar to the Finish Pass generated by a Flat Land Finishing cycle.

  When enabled, one or more outer passes, following the exact contour of the selected geometry including offsets and boundaries, will be created for each layer.

- **Undercut / Negative drafted walls**

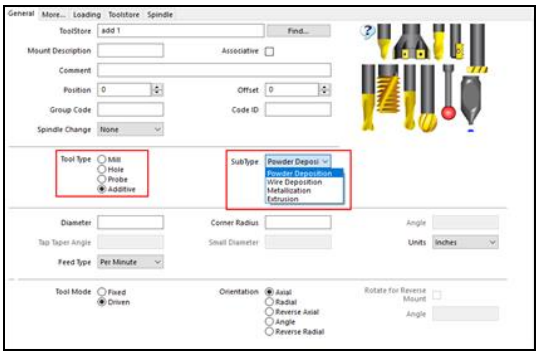
  To enable undercut shapes to be created, an **Undercut model** option has been included that will ensure that the cycle does not fill up the voids

- **Link control**

  Long links as **Clearance** will make the tool retract to clearance at rapid rate. Longs links as **Optimised** will make the tool retract by **Safe Distance** height, and travel at high feed rate to

the next point.

Additive Tool



For this release an Additive tool has been added to the toolchange:

- On the General Tab, set **Tool Type** to **Additive** and select one of the Additive **Sub Types**: **Powder Deposition**, **Wire Deposition**, **Metallization** or **Extrusion**.
- On the General Tab, **Diameter** and **Corner Radius** define the dimensions of the additive bead, where the corner radius is applied to the uppermost side; optionally, a **Flute Length** can be defined on the More tab.
- On the Loading tab, **Shank Length** is used to define the length of the mixing jet from the nozzle to the top of the bead, in the case of Powder Deposition. Overall, it is the distance between the nozzle and the region where the material deposits on the part.
- The More Tab includes additional technology additive parameters that can be used to control the additive process and can be directly output to the NC using Code Wizard.

New Wire Technologies

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

Mitsubishi

- FA10 BRD-B13W031-A10.
- FA10 BRD-B13W031-A13.
- FA20-V, BRD-B13W042-A8.
- FA30 BRD-B13W032-A10.
- MP1200 BRD-B13W135-A1.
- MP1200 BRD-B13W146-A5 AdvancePlus 2.
- MP1200 BRD-B13W146-A6 AdvancePlus 2.
- MP1200 BRD-B13W146-A7 AdvancePlus 2
- MP2400 BRD-B13W136-A1.
- MP2400 BRD-B13W147-A5 AdvancePlus 2.
- MP2400 BRD-B13W147-A6 AdvancePlus 2.
- MP2400 BRD-B13W147-A7 AdvancePlus 2.
- MP2400 DCUBES BRD-B13W164-A3 Advance V10.
- MP4800 BRD-B13W137-A3.
- MV1200R BRD-B13W118-A2.
- MV1200R BRD-B13W118-A6.
- MV1200R BRD-B13W142-A0 Advanceplus 2.
- MV1200R BRD-B13W150-A7 Advanceplus 3.
- MV1200R BRD-B13W150-A8 Advanceplus 3.
- MV1200R D-CUBES BRD-B13W159-A5.
- MV1200R D-CUBES BRD-B13W159-A7.
- MV1200S BRD-B13W117-A2.
- MV1200S BRD-B13W141-A0 Advance 2.
- MV1200S BRD-B13W149-A5 Advance 3.
- MV1200S BRD-B13W158-A4.
- MV2400R BRD-B13W152-A8 Advanceplus 3.
- MV2400S BRD-B13W119-A2.
- MV2400S BRD-B13W119-A6.
- MV2400S BRD-B13W143-A0 Advance 2.
- MV2400S D-CUBES BRD-B13W160-A4.
- MV2400S-DC_BRD-B13W160-A7.
- MX600 BRD-B13W123-A10.
- MX600 BRD-B13W123-A9.

Fanuc

- Robocut Alpha C4001A.
- Robocut Alpha C400IB.
- Robocut Alpha C600IB.
- Robocut Alpha C800IB.
- Robocut Alpha FANUC0ID.
- Robocut Alpha FANUC1ID.

Makino

- Makino technology has been updated to V33.

ACorange

- A technology database has been added for ACorange.

Change Notification Improvement for the Solid's File

Change Notification informs the user if a later version of the solid's file within the .ppf file is detected; this check is activated by the **Change Notification** setting on the **Solids** tab of the **Preferences** dialog.

In previous Edgecam versions, when opening a .ppf file containing a solid's file on an inaccessible path, there was a delay before Edgecam became active. The delay was caused by the change notification check not being able to locate the solid's file.

We have, therefore, changed the way that Change Notification works by disconnecting it from the File Open process and running it in the background. If it detects that the original solid's file is not accessible, it disables the Change Notification for the current Edgecam part and informs the user that the link to the original solid's file has been lost.

Performance improvements

Roughing Waveform calculation time

We have improved the Roughing Waveform calculation time in at least 15%, some cases up to 60%. We have improved the calculation time mainly for the known performance bottlenecks, such as usage of small stepover, narrow channels and heavily curved model regions.

Reduce regeneration on turning

Turning sequences will now cause less involuntary regeneration, especially in some specific commands like Angular Move.

Reduce prismatic geometry data

With Prismatic Geometry enabled, we only process the geometry on the model between the level and depth of the cycle. This can lead to improved performance on very complex components.

CAD Support Enhancements

Support for the latest CAD versions

The following CAD / file versions can now be loaded:

- SpaceClaim 19 (ACIS V28).
- Parasolid version 30.1.247.
- Inventor 2019.
- Designer 2019 R1 (*.vdf).
- Creo 5.0.

Note: In order to load Inventor 2019 files, the machine needs to have Inventor or Inventor View 2019 installed.

Cset IGES Loader Retired

The Cset Ci2x IGES loader used in Edgecam and Part Modeler has now been retired.

The IGES loader now defaults to using SolidLink and the redundant Cset modifiers have been removed from the associated Edgecam and Part Modeler dialogs.

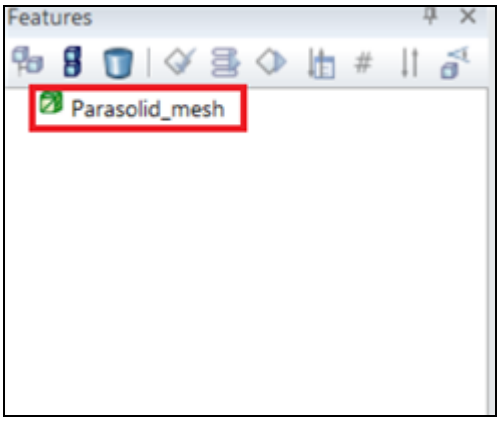
Load Designer files without requiring a Solid Machinist option license

It is now possible to load Designer files (*.v_t and *.vdf) into Edgecam without requiring a Solid Machinist option license:

- All Edgecam system licences allow Designer files (*.v_t and *.vdf) to be loaded.
- Feature Finding and Strategies are fully supported for these files on all CAM system licences.

Note: Other solid model files, including Parasolid, cannot be opened in Edgecam unless an appropriate Solid Machinist option license is available.

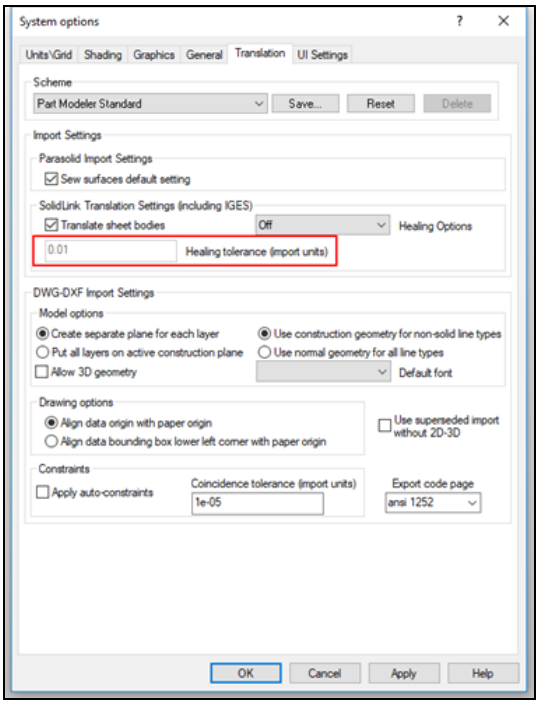
Load Parasolid Mesh from VDF files



For this release, Parasolid Mesh can be loaded from VDF files; these are triangulated bodies made up of free form faces:

- The Parasolid Mesh bodies are depicted by a green triangulated cube icon in the Feature and Setup Browsers to differentiate between the two types of Parasolid body.
- The Mesh can be machined directly; edges can be picked to define boundaries and the Mesh can also be used to define stocks and fixtures.
- They are unsuitable for Feature Finding. The Features Ribbon interface does not differentiate between a Parasolid Mesh and Standard Parasolid and, therefore, for Parasolid Mesh, the Feature Find commands cannot be hidden and running the commands will fail to find any features.

Part Modeler IGES Enhancements

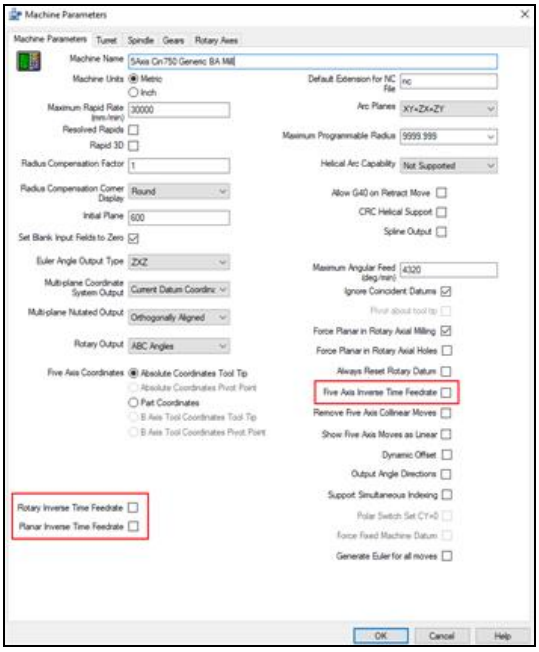


For this release, C-set has been removed and SolidLink will be used, by default, to import IGES files in Part Modeler:

- The 'IGES Import Standard Settings' have been removed.
- A **Healing tolerance** field has been added to the 'SolidLink Translation Settings'.

Code Wizard Enhancements

Inverse Time Feedrate extended support



Inverse Time Feedrate is a method of representing the velocity of a move based on cutting time instead of cutting feed.

Historically, it has been available for 5 Axis output but, in theory, it can be used on any movement provided that the controller supports it and the correct G-code is specified.

For this release, Inverse Time Feed can be used separately in three modes:

- 5 Axis (already existing).
- Planar mode.
- Rotary mode.

The post processor allows these to be configured, as required, in the Machine Parameters dialog.

In the NC-Style area, the **Inverse Time Feed G Code** has been moved into the G-Codes tab because it is no longer exclusive to 5 Axis.

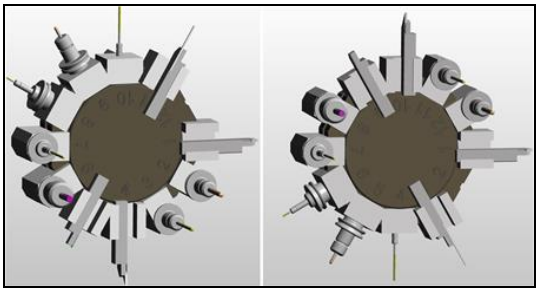
Note: Your post needs to be updated to the latest template to benefit from this improvement.

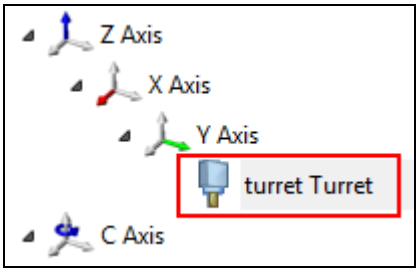
The **[FEEDMODEGCODE]** token also needs to be used appropriately in the respective code constructors.

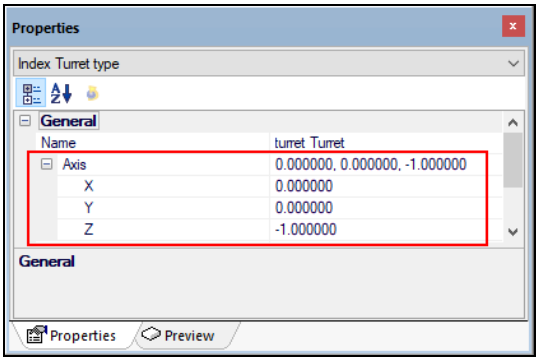
Some specific cycles will not output inverse time feeds, reverting to the appropriate feed mode G-code. These include:

- Hole cycles (all drilling, helical hole, thread milling).
- Turning cycles.

Reversing the Direction of Mounting







In previous versions, the order in which the tools were mounted could not be changed for a revolving turret. For this release, Edgecam includes this capability.

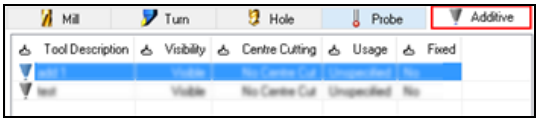
The user needs to change the Turret Axis, in the Machine Tree, in Code Wizard. Changing the Turret Axis, by reversing its vector definition, will change the tool position order (Clockwise / Anticlockwise ordering).

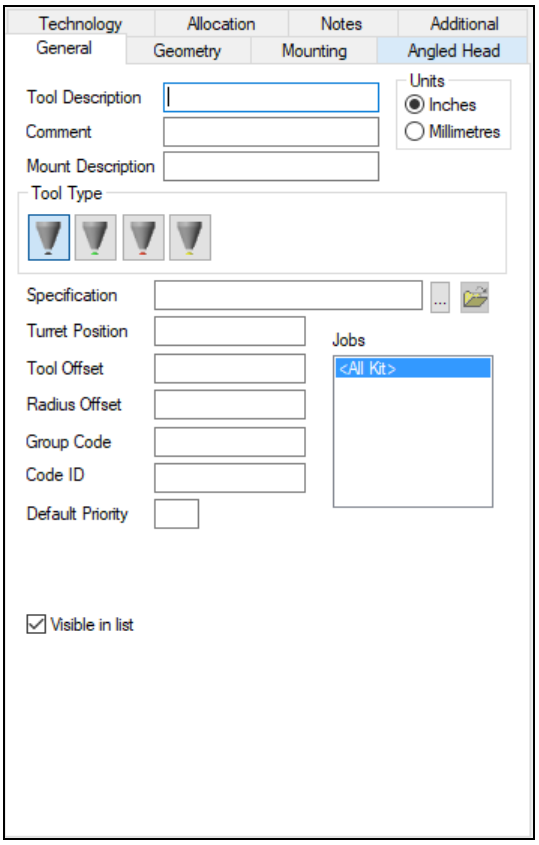
Euler Angles

The ability to output Euler angles for every move was previously restricted, and switchable via a PCI variable. This has now been fully implemented and is available via a "Generate Euler for all moves" machine parameter.

ToolStore Enhancements

Edgecam ToolStore - Additive / Hybrid machining





Additive Tools can now be created and stored in the ToolStore; these are listed under the **Additive** Tab.

There are 4 additive types:

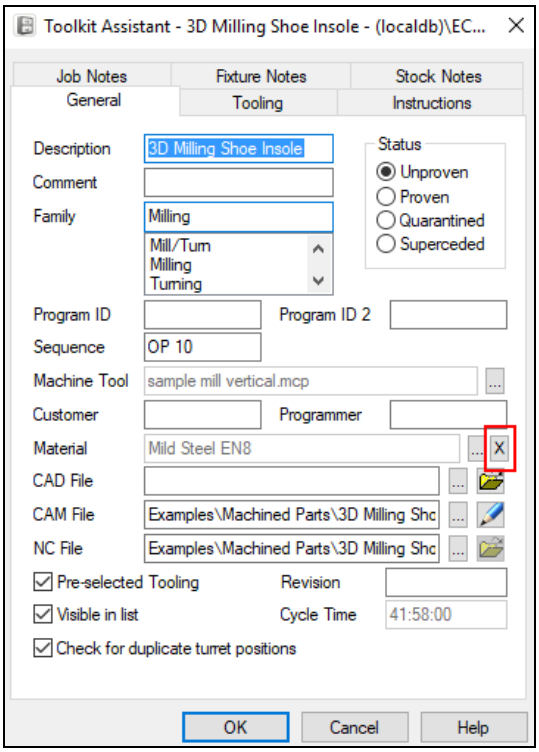
- Powder Deposition.
- Wire Deposition.
- Metallisation.
- Extrusion.

Edgecam has been primarily designed for Additive Powder.

The dimensions entered on the **Geometry** tab are for that of the additive bead; **Diameter**, **Corner Radius** and **Flute Length**. The **Shank Length** defines the length of the mixing jet from the nozzle to the top of the bead.

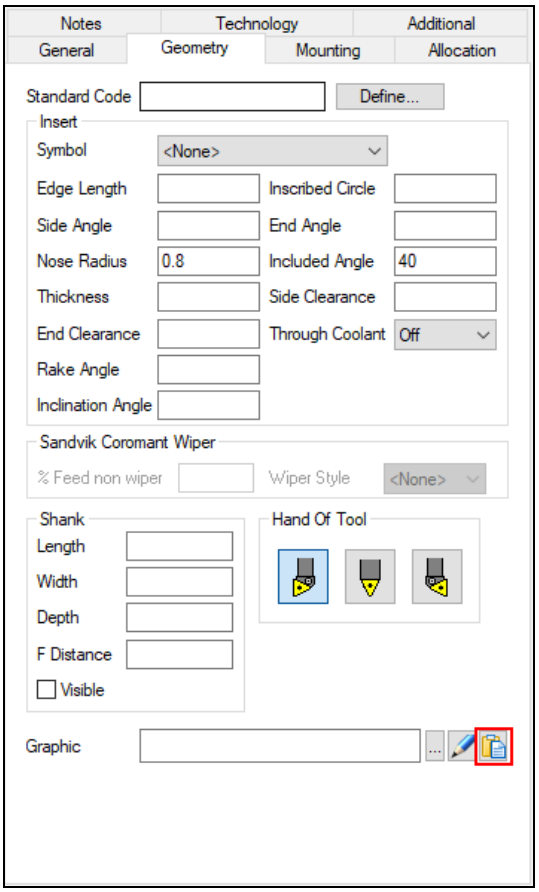
The **Technology** Tab stores information specific to the additive process.

Edgecam ToolStore - Removing Material setting from a Job/Toolkit



For this release, we have added a button to remove the Material setting from a Job/Toolkit.

Edgecam ToolStore - Paste Button implemented on Geometry tab for Turning Tools



A **Paste** button has been added to the Geometry tab for Turning tools which will enable solid insert graphics to be used; this was already available on the Mounting tab for pasting solid holders. The resultant .meg graphic is displayed in Edgecam and Simulator for a more realistic view of the tool. The Toolpath is calculated from the parametric insert definition and, therefore, care must be taken to match the parametric values to the solid insert shape.

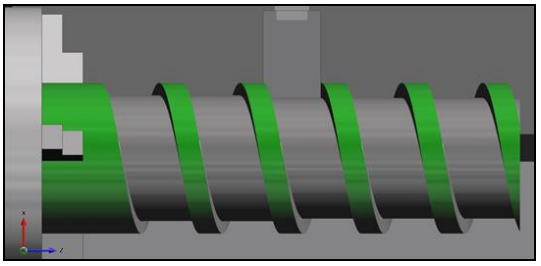
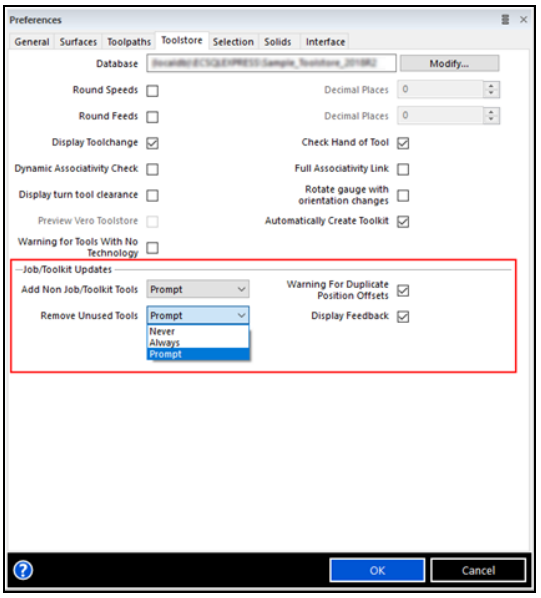
Maintenance Database Report

For a full list of maintenance items resolved in Edgecam 2019 R1, please refer to the **Maintenance Database Report**.

New Features in Version 2018 R2

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	
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.	

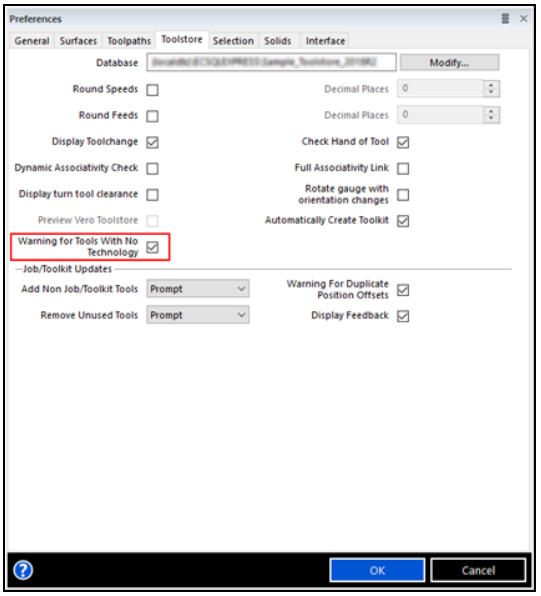
Preference added to output a warning when selecting tools without an associated technology

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.

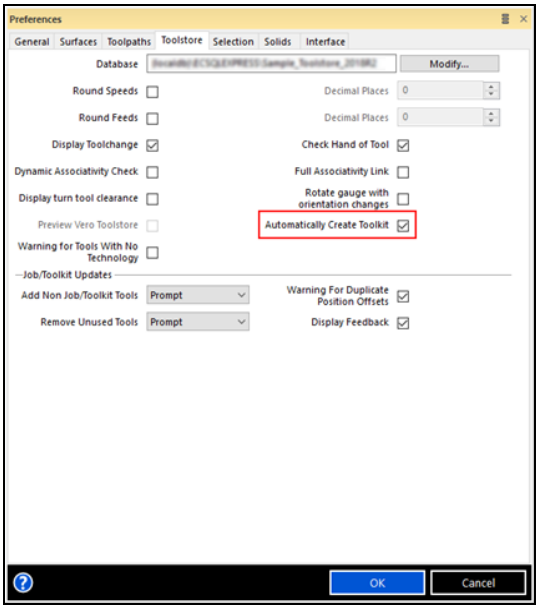


Option to not create a new Toolkit automatically

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.

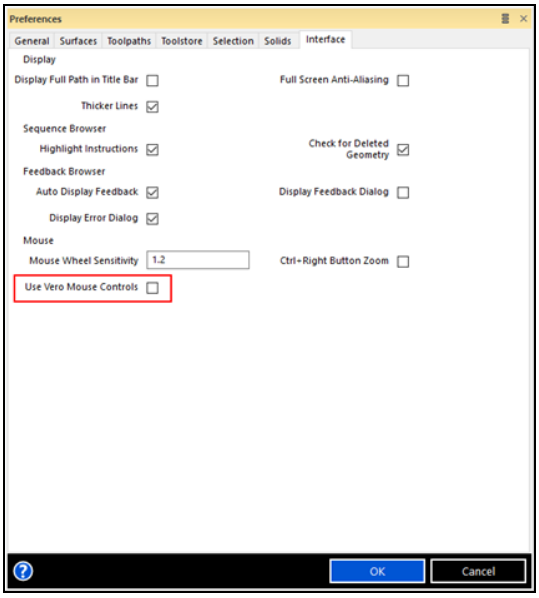
Note: To create a new Toolkit, type a name in the Toolkit field or click on one of the existing Toolkits to make a copy.



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.

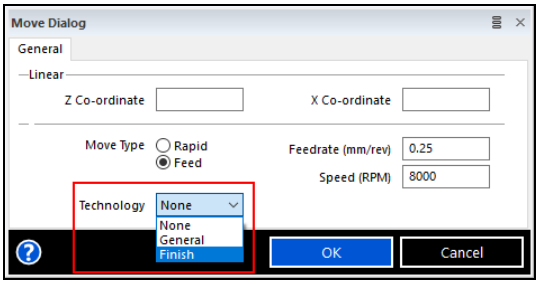
Selection (picking) methods

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.

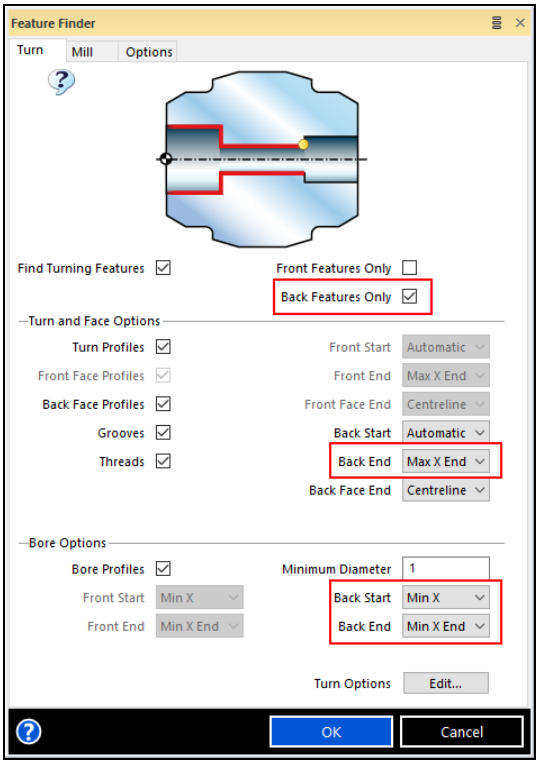
Note:

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.

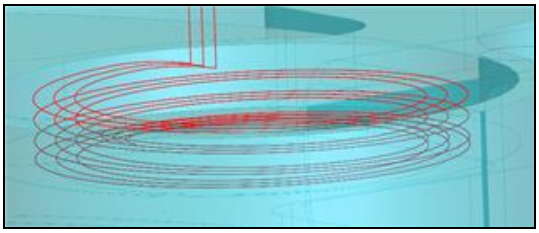


Profiling - Multiple XY Offset Passes with Helical

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

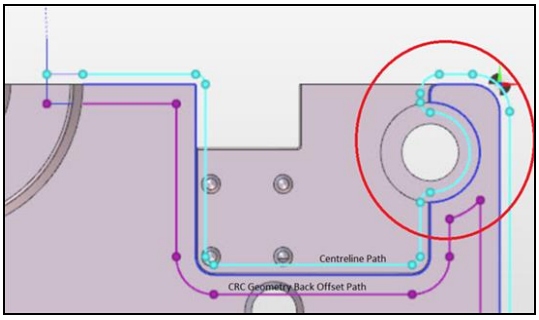
The combination of Profiling with a Taper tool with CRC **Compensation** = **Geometry** on certain geometries can lead to the back offset path intersecting with itself resulting in the back offset path having a different number of elements to the centreline path as shown in the image.

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).

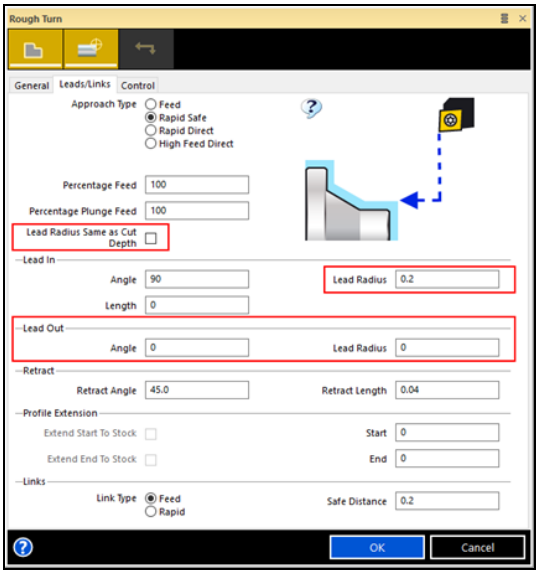
Rough Turn - Added Arc Lead On/Off

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 Sandvik Prime 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.



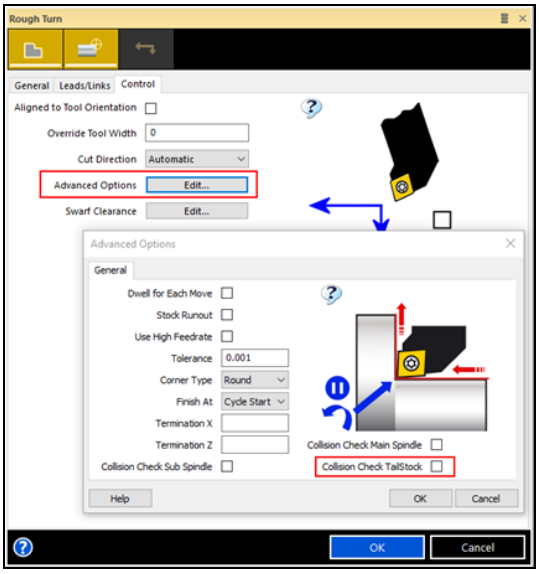
Rough Turn - Collision Avoidance for Tailstocks

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.

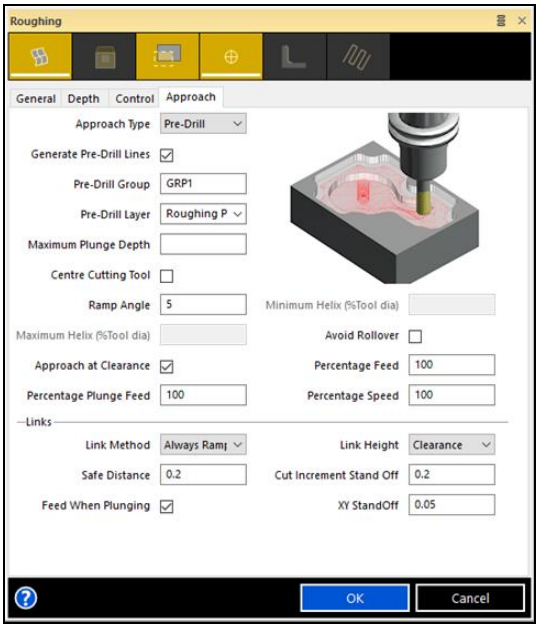
With **Approach Type** set to **Pre-Drill** and **Generate Pre-Drill Lines** selected, the Roughing cycle will plunge at the pre-drill locations assuming that the holes have already been drilled. A **Safe Distance** value must be specified to avoid making a rapid move into the rest material left by the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

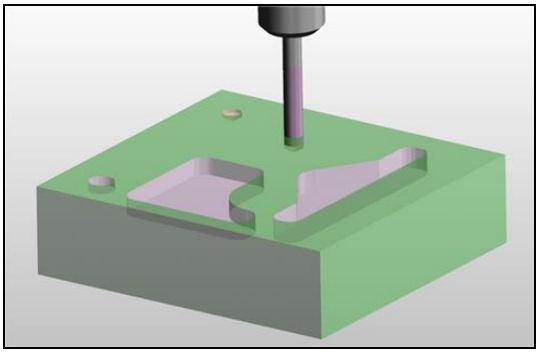
After the Roughing cycle has generated the pre-drill group of lines, the user can insert a Hole cycle prior to the Roughing to drill these; selecting a drill with a slightly bigger diameter than that used for roughing. In the Hole cycle:

- 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.

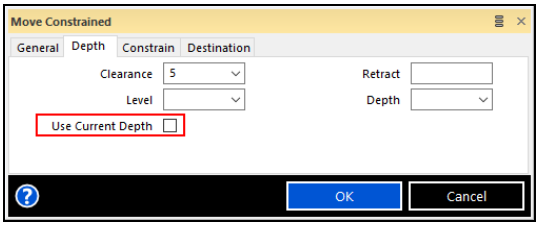


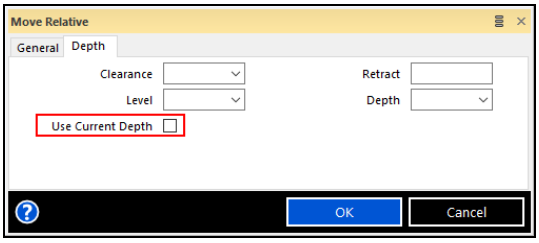


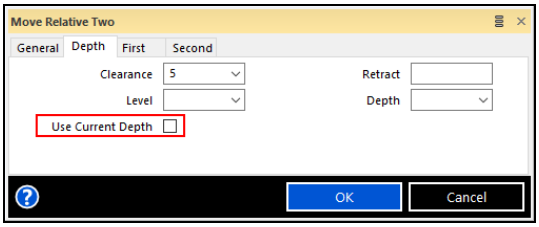
Move Constrained, Relative and Relative Two - Option added to Perform Moves at Current Depth

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.

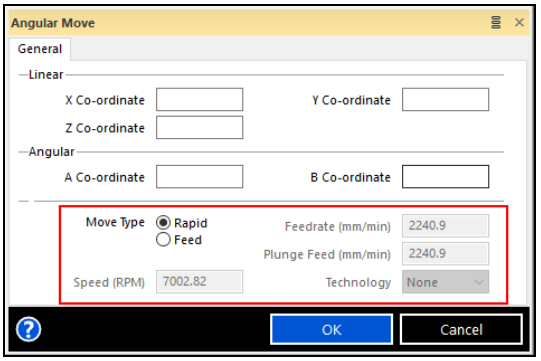
Thread Milling

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.

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.

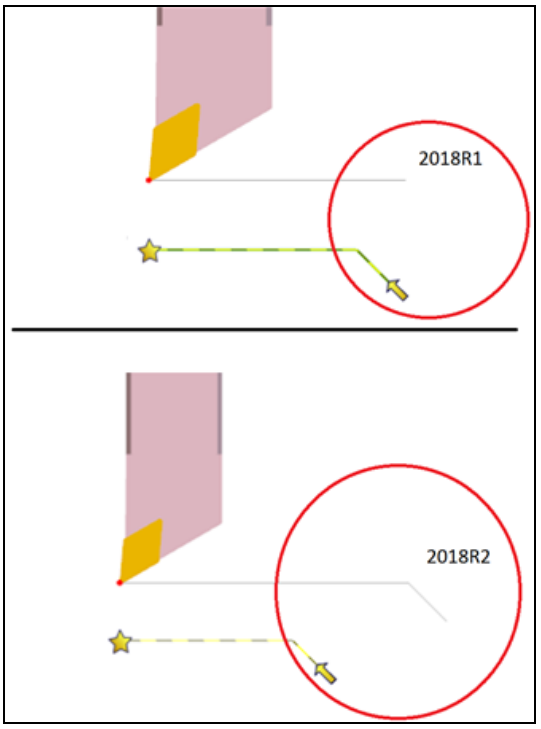
These modifiers are now unavailable when Rapid Mode is set and the NEXTSPEED look ahead in the code generator has been fixed to ensure that correct values are output to the NC.



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.

Mill Features from Wireframe

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

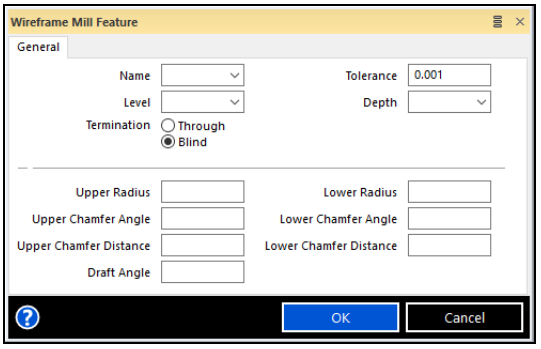
This command can be used when the faces or edges of a solid are not suitable to create the feature. The geometric input to the feature is always a 2D wireframe profile comprised of lines, arcs, etc:

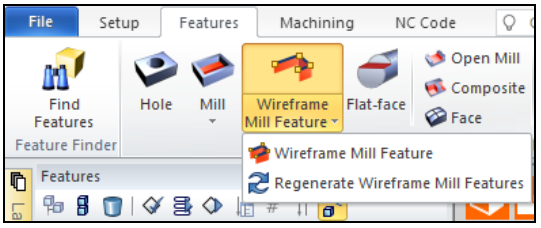
- The level and depth of the feature are specified on the dialog. Additional modifiers are available for defining an upper/lower radius or chamfer and draft wall angle.
- When the Feature is edited, additional parameters will be available allowing you to change the Coordinate Input, Reverse the Feature 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.

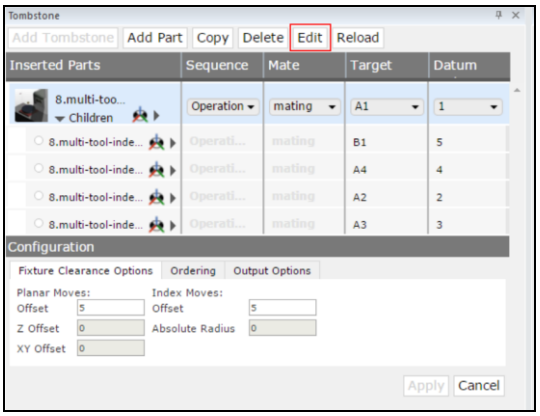
In this release, the module has been enhanced and the first step is now to insert the Tombstone ('seed') file. Only after this is done will the user be able to continue with the assembly. This also forces a **Save as** which prevents the accidental overwriting of the precious 'seed' file.

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:

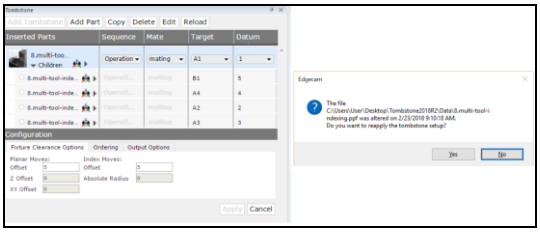
- The part needs to be available in the original location (at the time of insertion into Tombstone), otherwise, it cannot be edited.
- Children cannot be edited because they are instances of a Parent part which can be edited.
- To edit a part, select the Parent in the Browser which then enables the **Edit**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

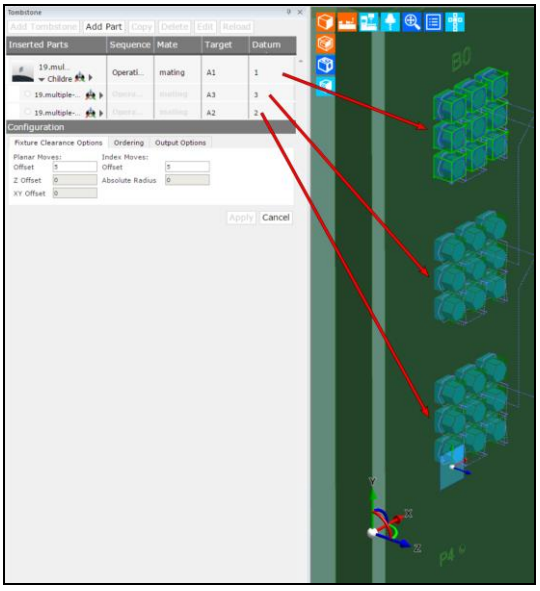
Unlimited Number of Parts to Insert

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

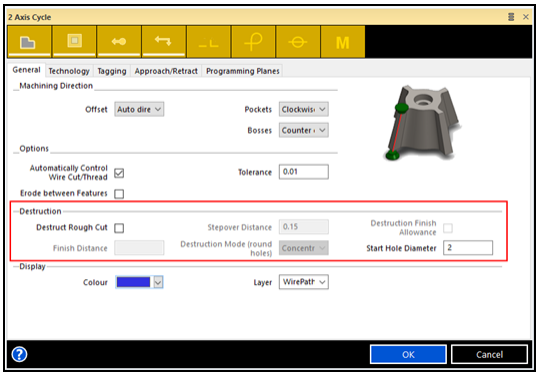
- BRD-B13W032-A13,FA30 (V2.0)
- BRD-B13W062-A8,FA-VS (V10.0)
- BRD-B13W111-A0,NA1200 (V1.0)
- BRD-B13W151-A5,MV2400S_ADVANCE3 (V3.0)
- BRD-B13W158-A3,MV1200S_D-CUBES (V1.3)
- BRD-B13W159-A3,MV1200R_D-CUBES (V1.3)
- BRD-B13W159-A4,MV1200R_D-CUBES (V1.4)
- BRD-B13W160-A3,MV2400S_D-CUBES (V1.3)
- BRD-B13W161-A2,MV2400R_D-CUBES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



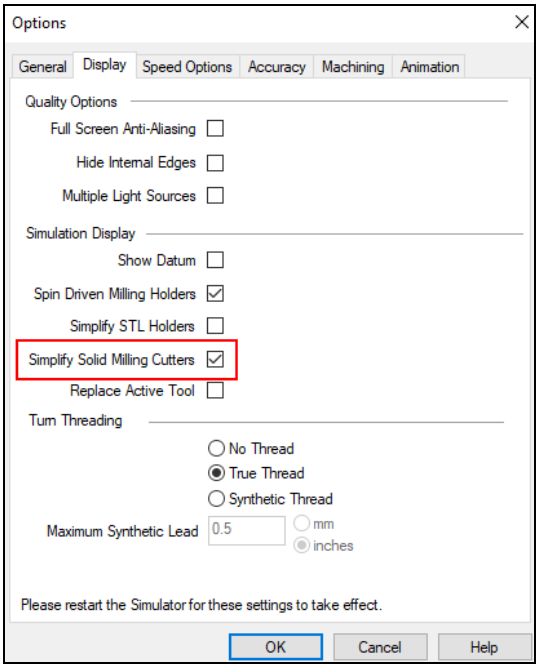
Notes:

- 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.

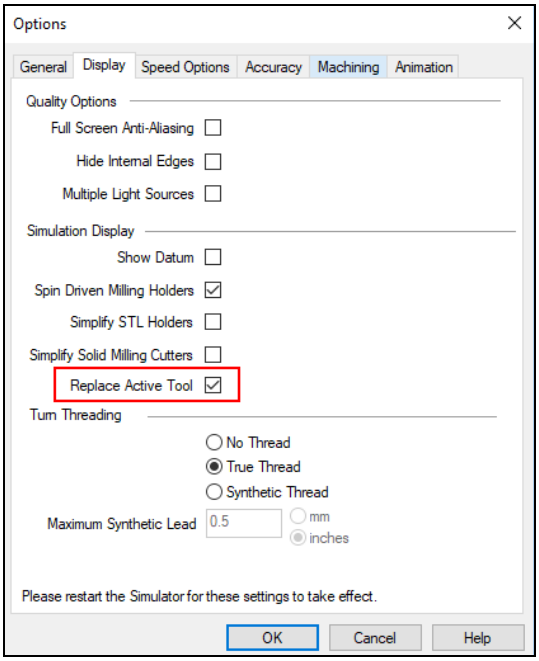
Replace Active Tool in Simulator for Manual Toolchange

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

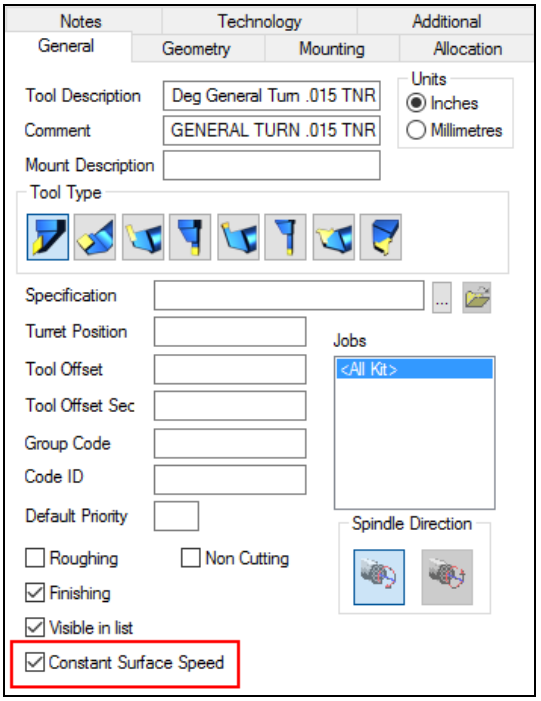
Edgecam ToolStore - Constant Surface Speed option added for Turning Tools

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.

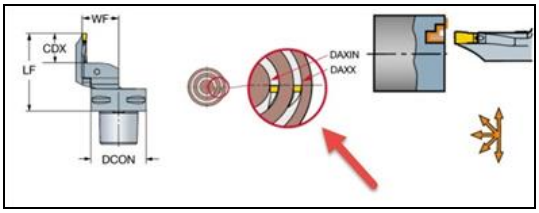


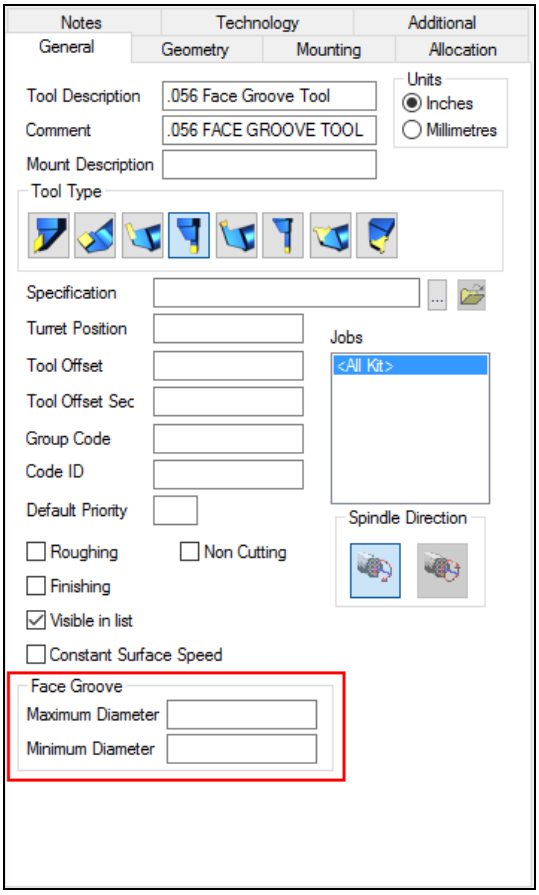
Edgecam ToolStore - Minimum and Maximum Diameter Range modifiers added for Face Groove Tools

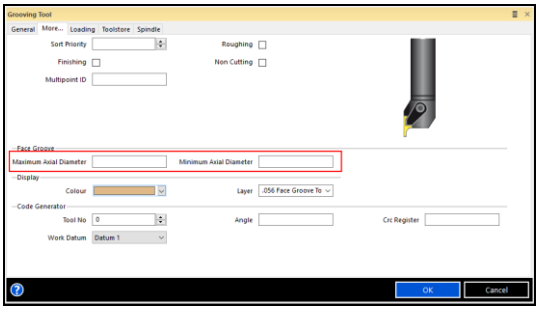
Minimum Diameter and **Maximum Diameter** range modifiers have been added for Face Groove Tools; they do not change the geometry of the tool.

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

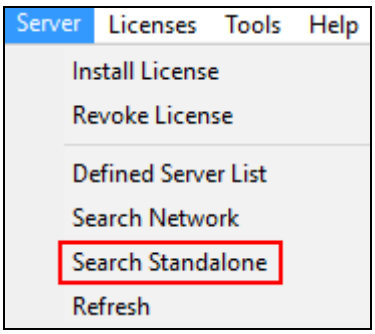
The Sentinel RMS software that our CLS licensing is based on has been upgraded to version 9.2.1.

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

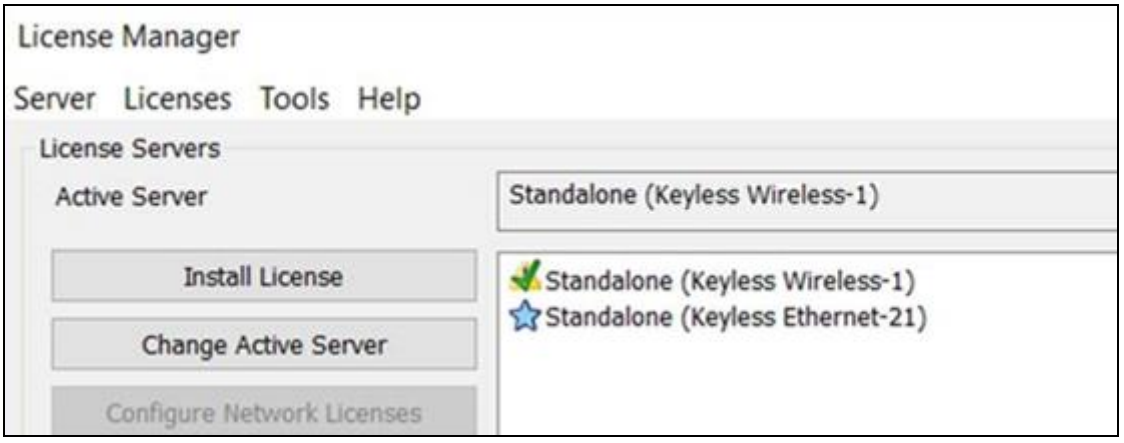
We have introduced the new **Search Standalone** option in the License Manager Server menu to update the list of options in License Manager, for example, when a key has been added to the system. License Wizard will also list all of the Standalone options.



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.

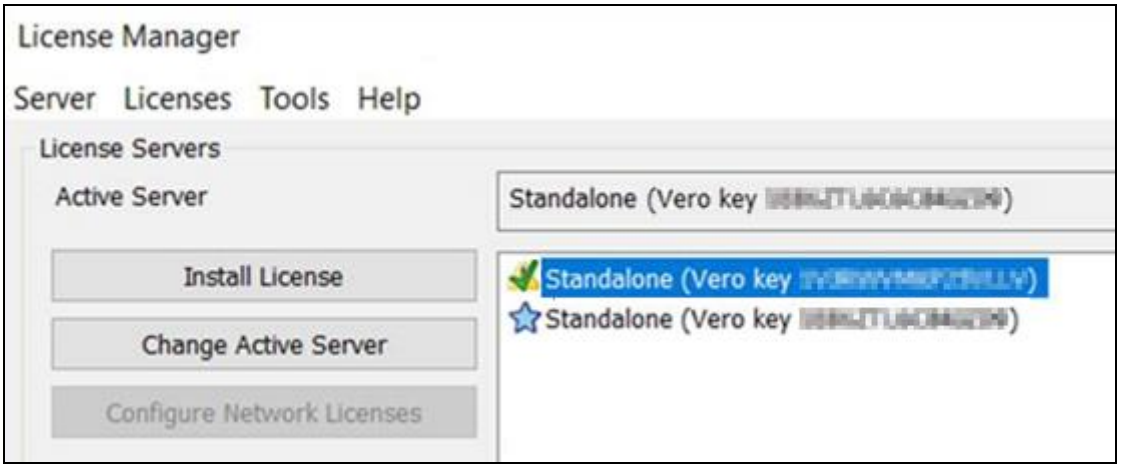
Locking Standalone licenses to Wireless and Ethernet options

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