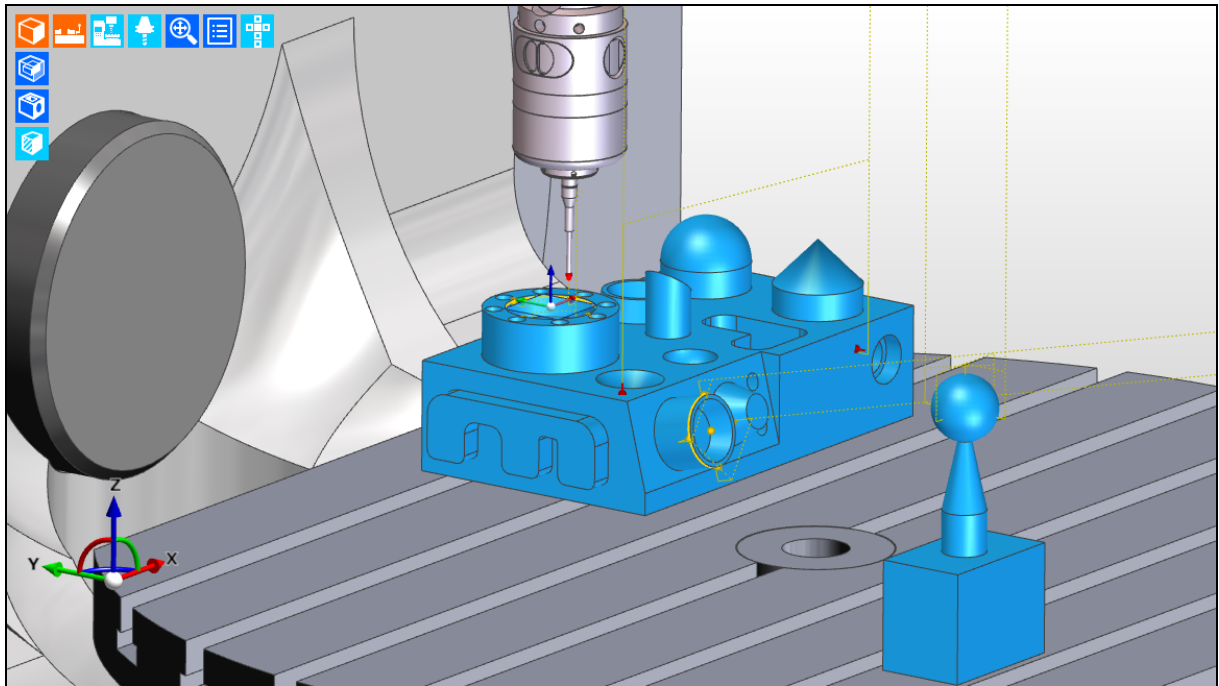


# What's New in Edgecam 2018 R1



edgecam

This document highlights new product features and enhancements in Edgecam 2018 R1.

**To run Edgecam and Part Modeler 2018 R1, the expiry date in the license must be September 2017 or later.**

# Contents

---

'WHAT'S NEW' DOCUMENT OVERVIEW .....	4
IMPORTANT INFORMATION .....	5

## Manufacture Enhancements

ROUGHING CYCLE - DETECT UNDERCUT STOCK ADDED TO GENERAL TAB .....	6
ROUGHING CYCLE - WAVEFORM - ABILITY TO SET HELIX DIAMETER ON APPROACH .....	7
PASS FEATURE BOUNDARY TO CYCLES.....	7
B AXIS CONTOURING CYCLE - MAXIMUM ANGULAR INCREMENT .....	9
ROUGH TURN - COLLISION CHECK CHUCKS .....	9
ROUGH TURN - EXTEND PAST STOCK EDGE .....	10
PROFILING - SPRING CUTS ADDED TO GENERAL TAB.....	10
PROFILING - ADJUST FEEDRATE ON ARCS ADDED TO GENERAL TAB .....	11
PROFILING - UNDERCUT PROFILING WITH PROTECT SOLID IS NOT PERMITTED.....	11
PRESERVING FEATURES THAT WERE FOUND ON A STOCK SOLID.....	12
SPEED AND FEED MODIFIERS ARE NOW LOCKED WHEN USING TECHNOLOGY VALUES FROM THE TOOLSTORE.....	12
DEEP HOLE DRILLING .....	13
SPINDLE CONTROL COMMAND NOW CONTROLS THROUGH TOOL COOLANT .....	14
MOVE TO TOOLCHANGE / HOME ON MACHINES WITH ROTARY HEADS .....	15
SUPPORT FOR JTOPEX FILE FORMAT.....	16
HANDLING OF NESTED FEATURES FOR IMPROVED STRATEGY USE .....	16

## Interface Enhancements

SCREEN CAPTURE .....	17
IMAGES ON DIALOGS .....	18
NEW OPENGL DATUM .....	19
PDI API ENHANCEMENTS.....	19
FORMULAS IN DIALOGS .....	20

## Simulator Enhancements

SPINNING EFFECT ON HOLDERS DURING SIMULATION .....	21
----------------------------------------------------	----

## Code Wizard Enhancements

EULER ANGLES FOR MULTI-PLANE MACHINING .....	22
----------------------------------------------	----

**Code Generator Enhancements**

SYSTEM VARIABLE FOR EULER ANGLES ..... 23

**Strategy Manager Enhancements**

STRATEGY MANAGER - ISFEATUREFINISHED FUNCTION FOR HOLES ..... 24

**Important Licensing Changes**

SUPPORT FOR EDGE CAM LEGACY SENTINEL AND HASP KEYS REMOVED ..... 25

SENTINEL RMS UPGRADE TO VERSION 9.1 ..... 25

HOW TO ACCESS LICENSING IN 2018 R1 ..... 26

NEW LICENSING WIZARD ..... 27

MAINTENANCE DATABASE REPORT ..... 28

NEW FEATURES IN VERSION 2017 R2..... 29

# '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 2017 R2](#).

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

# Important Information

---

## CLS Licensing Changes

Significant changes have been made to CLS in this release to improve the new user experience, meet the requirements of multiple brands, improve the development process and make it easier to support. See [Important Licensing Changes](#).

## 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.

## ToolStore Server Installation

The 2018 R1 ToolStore Server will be installed in 'Program Files' rather than 'Program Files (x86)'.

## Designer

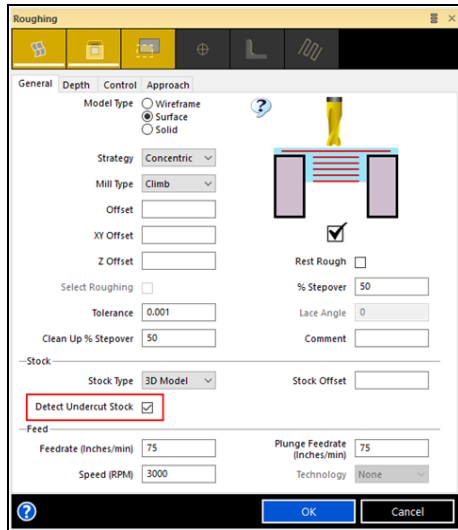
Designer will be released at 2018 R1 and will be listed on the DVD StartHere menu as a download.

## C/C++ - PDI's Retirement

C/C++ PDI's support in Edgecam is being phased out and, from the 2018 R2 release in April 2018, will not function. It is recommended that all new applications are developed as .NET plugins.

# Manufacture Enhancements

## Roughing Cycle - Detect Undercut Stock added to General tab

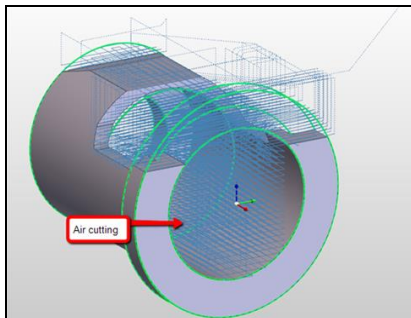


A new **Detect Undercut Stock** field has been added to the General tab of the Roughing cycle. This allows you to analyse the current stock and avoid areas that have no stock:

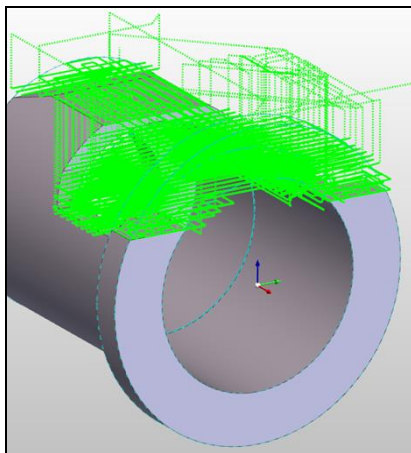
- This option should help to avoid air cutting and reduce the machining time. For this example, the machining time went from 2h 46min to 1h 16min.
- The option is an extra calculation step and could add time to the cycle generation time which would not be beneficial on complex parts with no undercut.

**Note:** Available on Standard Milling and Standard Production licenses.

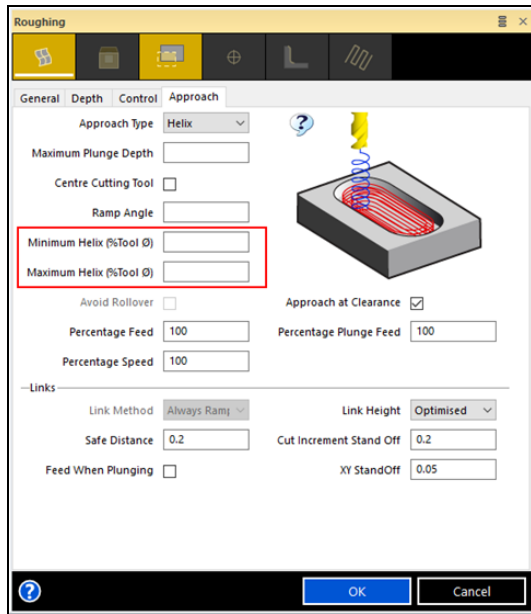
**Option not set:**



**Option set:**



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

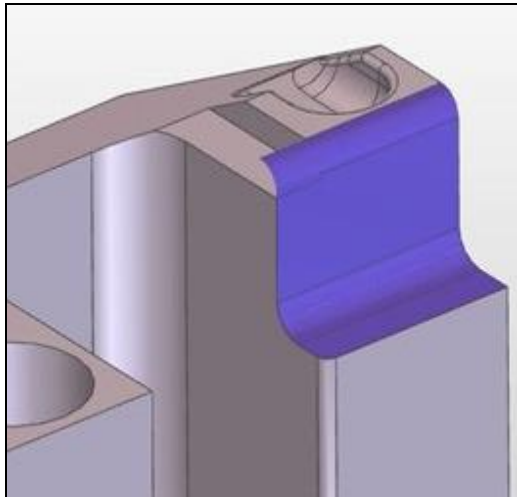


For this release, it is possible to define the Helix Diameter on Approach with **Maximum Helix (%Tool Ø)** and **Minimum Helix (%Tool Ø)** when using Waveform strategy for Roughing. Previously, the diameter was automatically set based on the size of the pocket, diameter of tool and whether or not it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

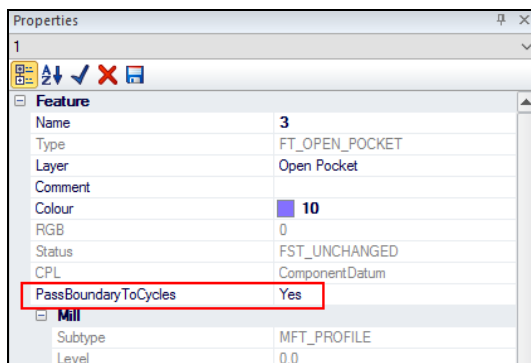
- The **Maximum Helix (%Tool Ø)** should be used unless limited by the pocket size.
- Equal values for both modifiers will define a fixed diameter for helical approaches.

## 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.

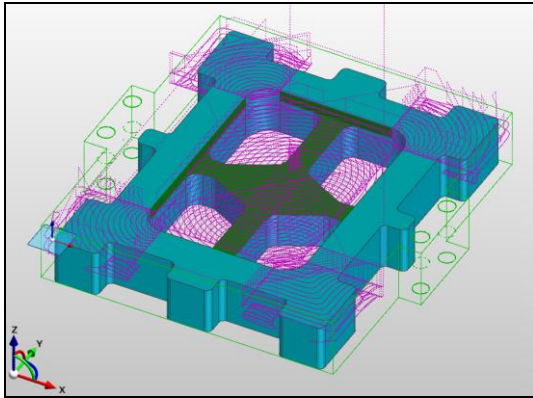
For this release, we have added an additional **Pass Boundary To Cycles** property to a group of features which will prevent this unnecessary toolpath from being created.



### 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**.

### Pass Boundary To Cycles = No

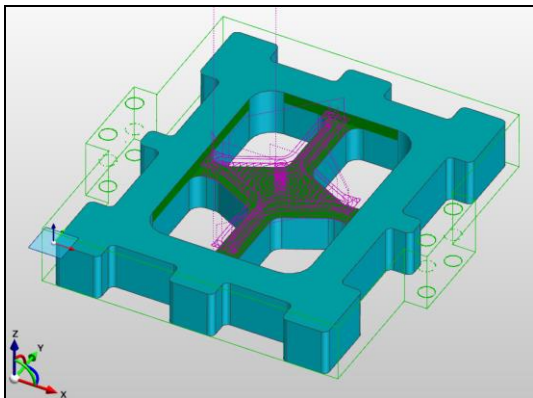


### Cycle Behaviour

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.

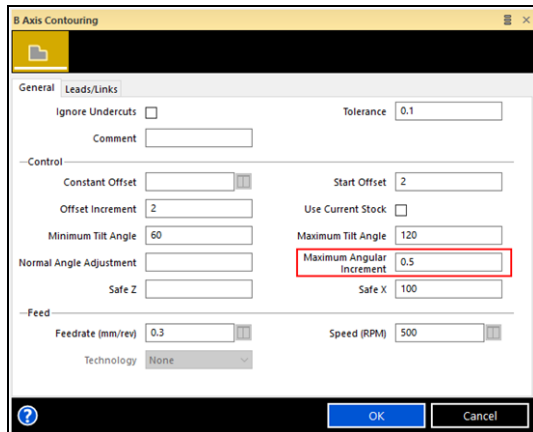
**Note:** The boundaries are generated by extracting an outer silhouette from the bottom face of the feature including any radius that might be connected to the wall.

### Pass Boundary To Cycles = Yes



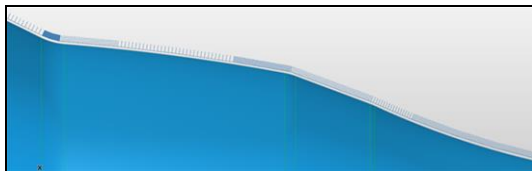
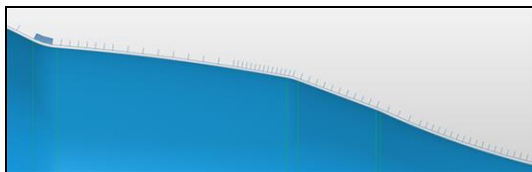


## B Axis Contouring cycle - Maximum Angular Increment



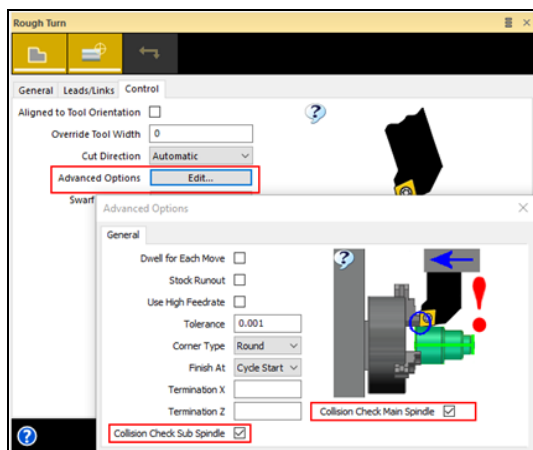
A new **Maximum Angular Increment** modifier has been added to the General tab of the cycle dialog:

- 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).

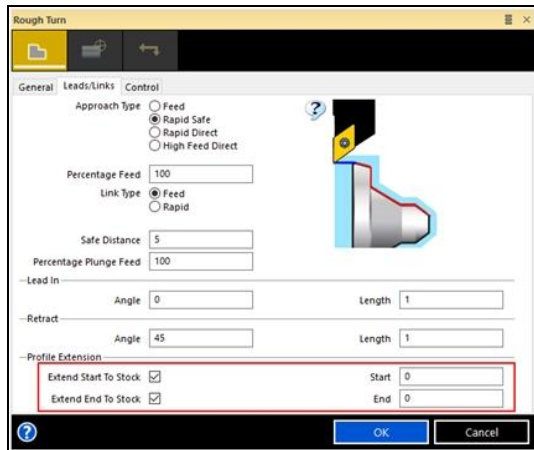
It uses the **Fixture Offset** in Update Fixtures (General tab) to determine the safety margin.

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.

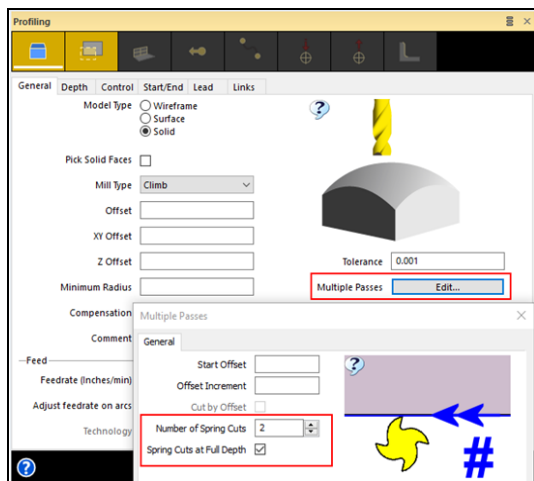
## Rough Turn - Extend Past Stock Edge



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.

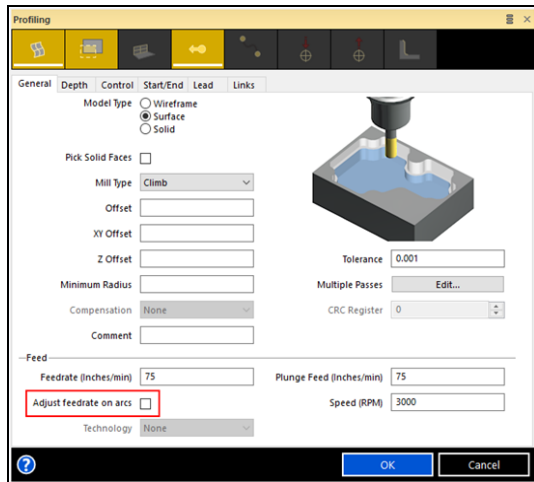
When profiling, tool deflection can result in the part being cut oversize, typically when machining hard materials. It can be necessary to repeat the profiling passes to achieve the correct size and surface finish.

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 Edgcam cycle time.

By making the adjustment in the cycle, Edgcam 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.

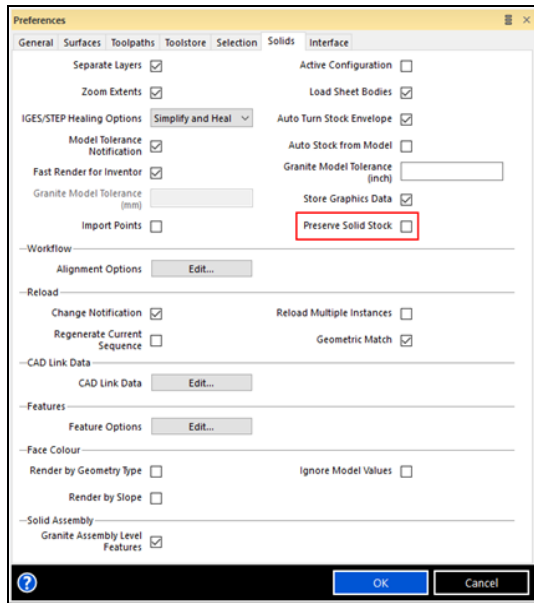
## Profiling - Undercut Profiling with Protect Solid is not permitted

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



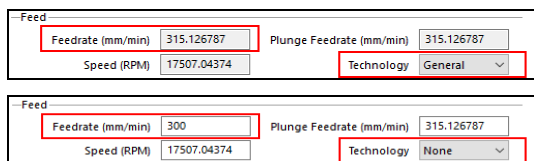
When a solid model is used as stock, you can still find features on it which can be used for cycle creation or for referencing, for example, with 'Move Relative To'.

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.

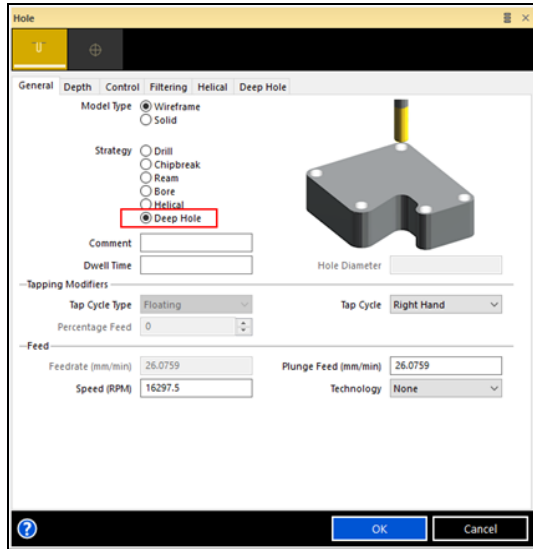
## Speed and Feed Modifiers are now locked when using Technology values from the ToolStore



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

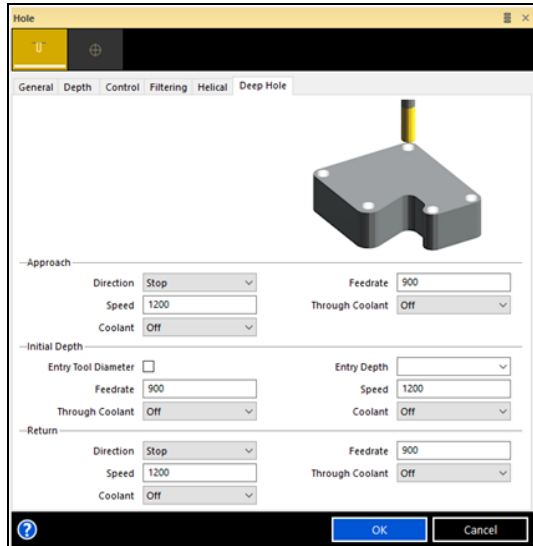


The Hole Cycle has been enhanced with the capability to perform deep hole operations where the user can control key parameters, such as tool direction, speed, feed, coolant and through tool coolant, on different sections of the operations.

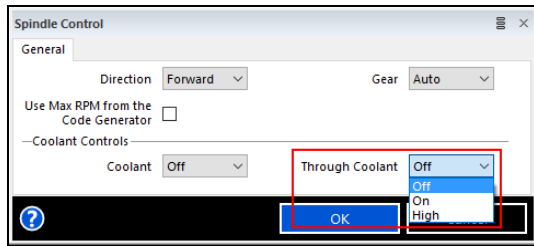
This is particularly important where specialised and rather fragile tools are used and the process requires accurate and precise output of such cycle elements.

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

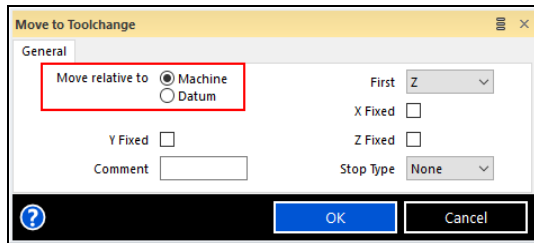


The Spindle Control command now controls Through Tool Coolant.

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.

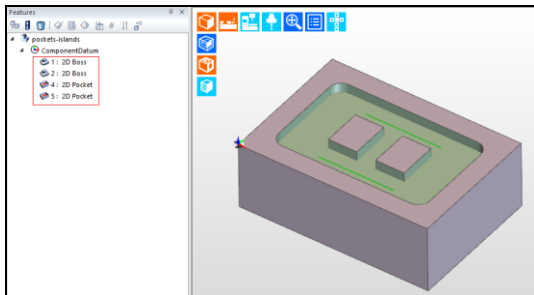
The JTOpen file is a lightweight 3D model format developed by Siemens PLM Software; designed as an open, high-performance, compact, persistent storage format for product data; used for product visualisation, collaboration, and CAD data sharing.

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:

- A Solid Machinist for Parasolid license is required to load JTOpen files.
- Edgecam only loads JTOpen files that contain solids.
- PMI data is not supported.
- Versions up to 9.5 are supported.

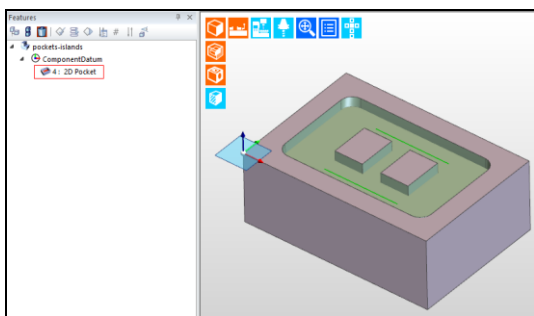
## Handling of Nested features for improved strategy use



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:

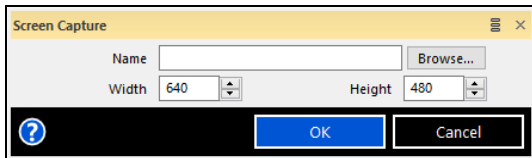
If compatible, only the Nested feature will be maintained and the others will be sent to the bin.



# 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.

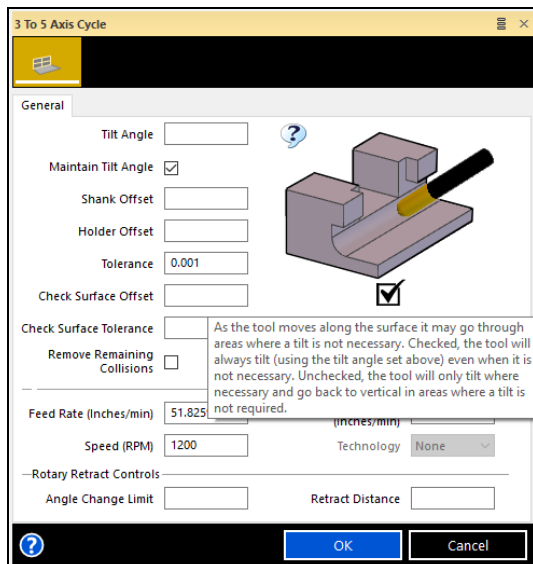
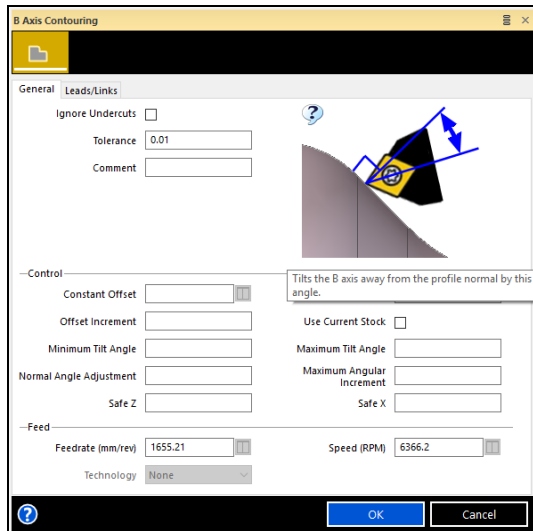
The Screen Capture command can be accessed within Edgecam using one of the following methods:

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

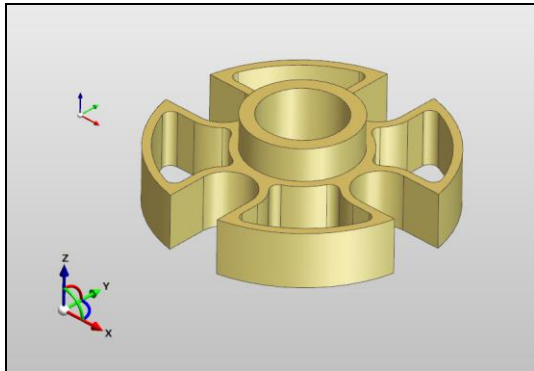
## Images on Dialogs

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.

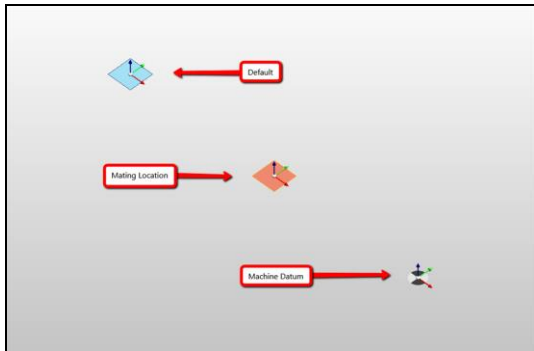


## New OpenGL Datum



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.

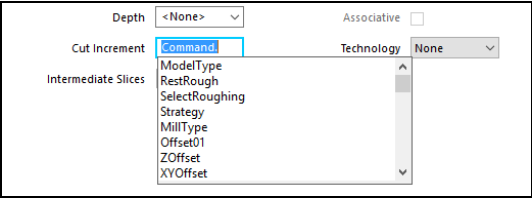


## PDI API Enhancements

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

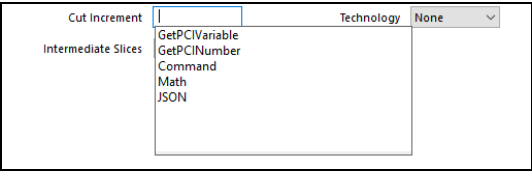


Following the work to implement Formulas in Dialogs for the 2017 R2 release, a number of enhancements have been implemented for this release which will help users to access the formulas and identify when they have been used:

### 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.

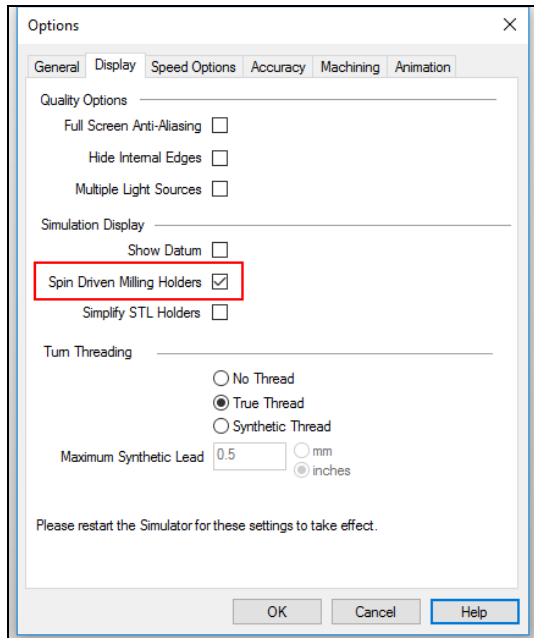


### Highlighting variables

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 Edgcam 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

Machine Parameters

Machine Name: ISO 3 Axis Mill

Machine Units: ☒ Metric ☐ Inch

Maximum Rapid Rate (mm/min): 30000

Resolved Rapids: ☐

Rapid 3D: ☐

Radius Compensation Factor: 1.1

Radius Compensation Corner Display: Round

Initial Plane: 600

Set Blank Input Fields to Zero: ☒

Euler Angle Output Type: ZXZ

Multi-plane Coordinate System Output: XYZ, YZY, ZYZ, XZX, YXY, XYZ, YZX, ZXY, XZY, ZYX, YXZ

Multi-plane Nutated Output

Rotary Output

Five Axis Coordinates

☐ Part Coordinates

☐ B Axis Tool Coordinates Tool Tip

☐ B Axis Tool Coordinates Pivot Point

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 Edgcam 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:

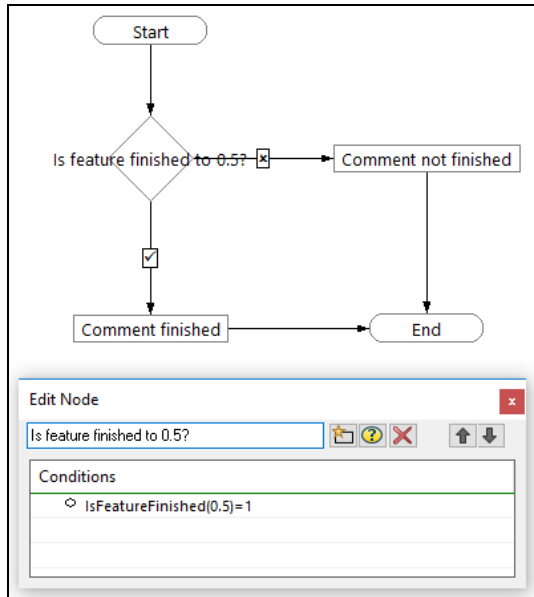
- Milling Macro Reference (MACRO 43 - INDEX PALLET).

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.



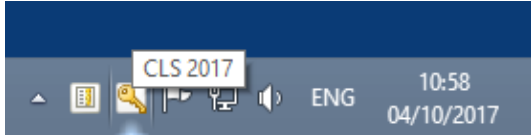
For customers who have a Sentinel RMS Network license, you must ensure that your **Sentinel RMS License Manager** is version 9.1 or higher. After installing Edgecam 2018 R1 and attempting to use network licenses, if the Sentinel RMS License Manager is earlier than version 9.1, the message shown below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

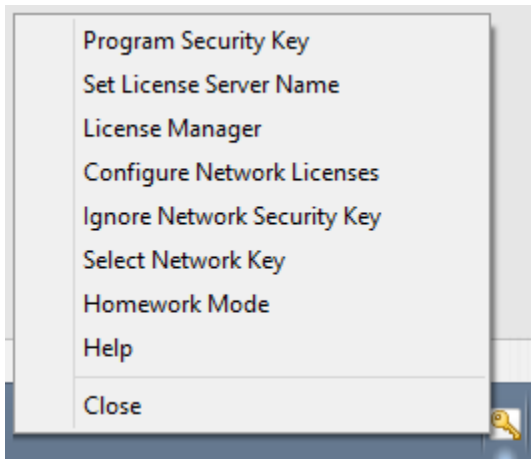
## How to access licensing in 2018 R1

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.

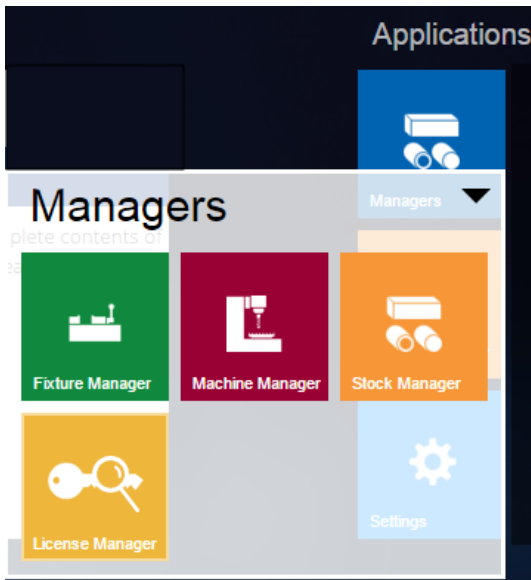
The License Manager, Configure Network Licenses, Homework Mode, Preview and Help menu options that were previously displayed on the CLS menu are all now in the License Manager.



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.



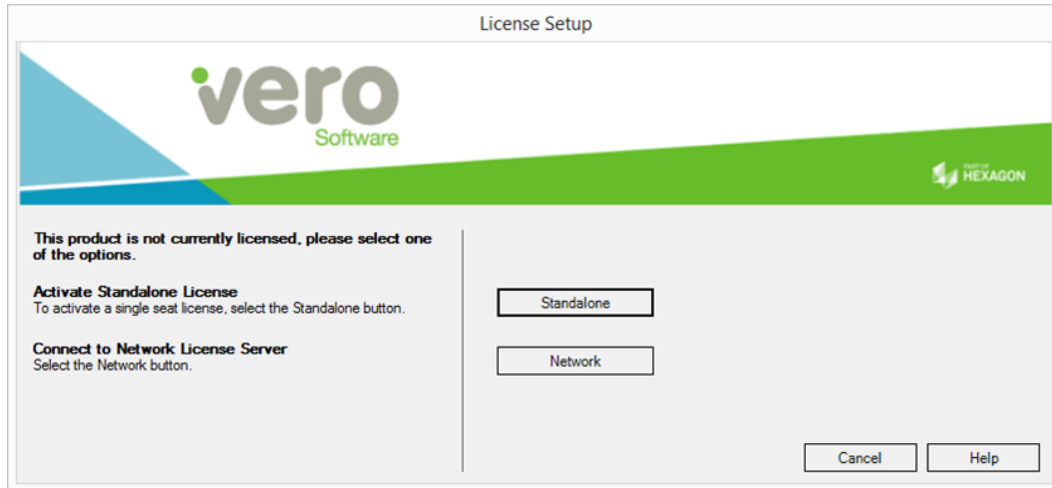
The Configure Network licenses option is accessed from a button on the License Manager dialog.

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.

# Maintenance Database Report

---

For a full list of maintenance items resolved in Edgecam 2018 R1, please refer to the [Maintenance Database Report](#).

# New Features in Version 2017 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

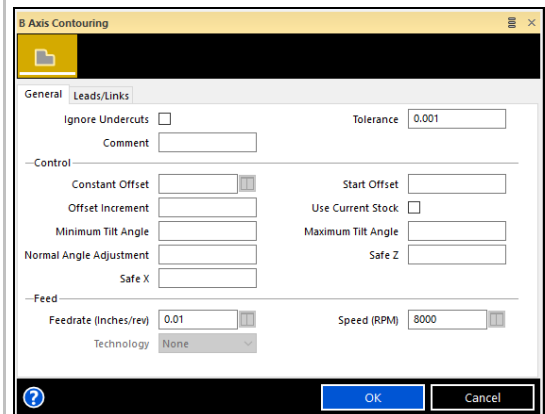
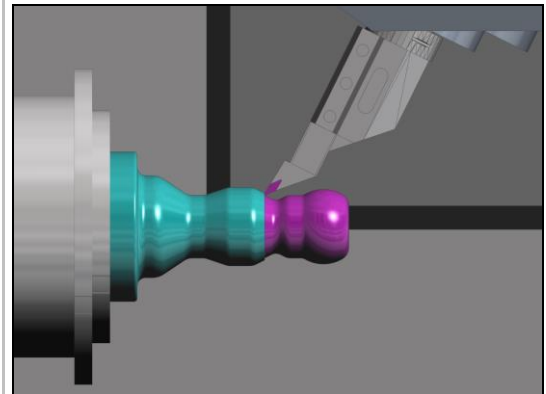
### B Axis Contouring Cycle

The B Axis Contouring Cycle has been introduced for this release.

This cycle is for use on turning centres with a programmable B axis.

This cycle rotates the B axis during profile turning keeping the tool normal to work piece; this helps to achieve a good surface finish while allowing the tool to get into undercut regions reducing the need for specialized tooling.

**Note:** To use this cycle, you need the Ultimate Turning or Ultimate Production Licence.

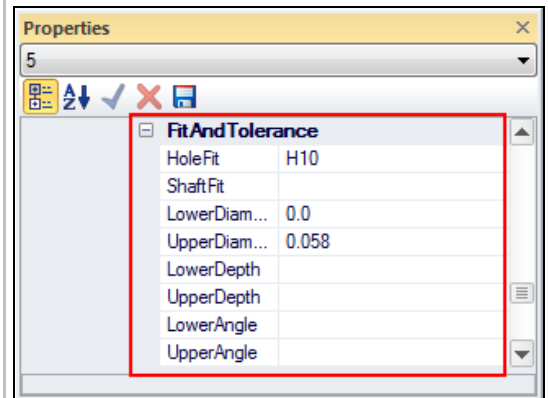
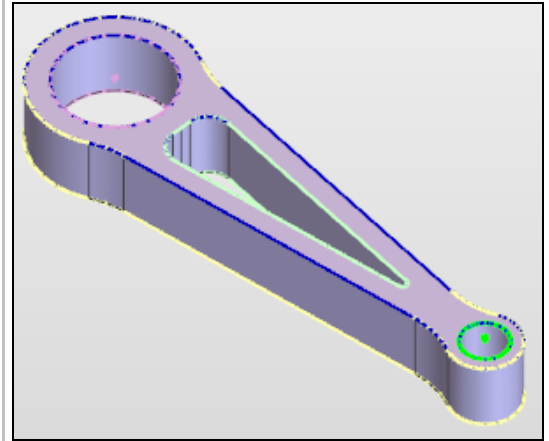
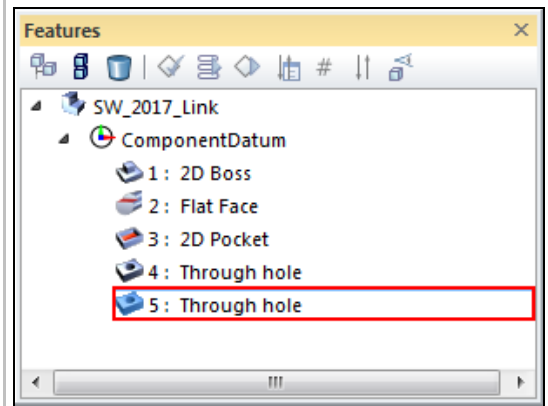


### Importing Limits / Fits for Hole Features

Standard hole fit definition and associated tolerances can now be extracted from SOLIDWORKS, INVENTOR and CREO/Pro-E models and displayed in Edgecam. This information is extracted via CADLinks and allows the user to apply the usual Edgecam functionality and machine the part to the specification required.

The holes must be defined in SOLIDWORKS, INVENTOR or CREO/Pro-E using the Hole command. Using the CADLinks button, the part is sent to Edgecam and, once automatic feature find has been completed on the part, any standard (e.g. H7) and tolerances (+0.021/-0.00) applied to the hole feature will be found by Edgecam.

**Note:** It is currently only possible to extract limit and fit information from hole features. The solid model must have the limit and fit information applied through the Hole command within SOLIDWORKS, INVENTOR or CREO/Pro-E.

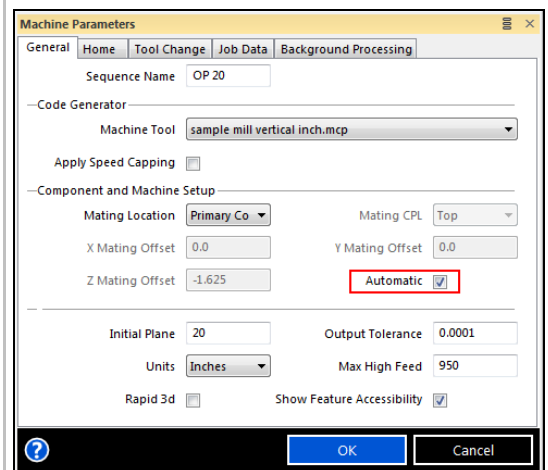


### Mating Position updates automatically when Stock is edited or created

For this release, we have added an **Automatic** option to the **General** tab of the Machine parameters dialog. This enables the Mating position to be updated automatically when stock is edited or created.

- When checked, the mating distances (**Mating CPL** and **X/Y/Z Mating Offset**) will be derived from the **Automatic** setting of the sequence setup.
- When unchecked, the mating distances can be manually adjusted. Checking **Automatic** again will reapply the derived mating distances.

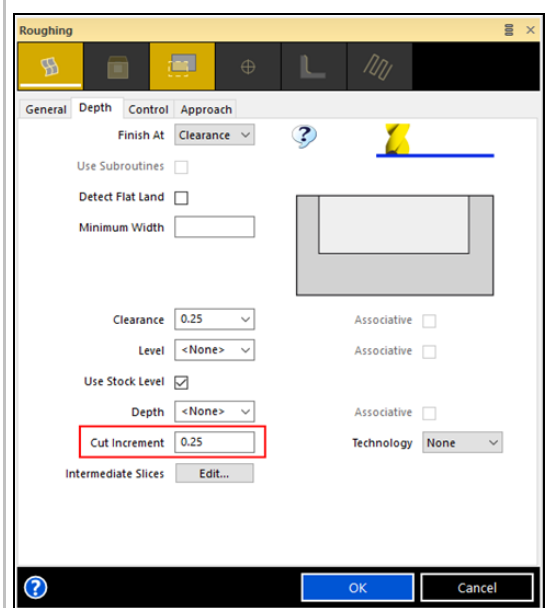
**Note:** This option is only available for Mill and MTM machines.



### Roughing Cycle - Blank Cut Increment allowed

For this release, you can leave the **Cut Increment** field on the Depth tab blank to produce a single pass at depth.

In previous versions, an error was issued because a **Cut Increment** value had to be specified. Roughing now behaves in the same manner as Profiling in this respect.



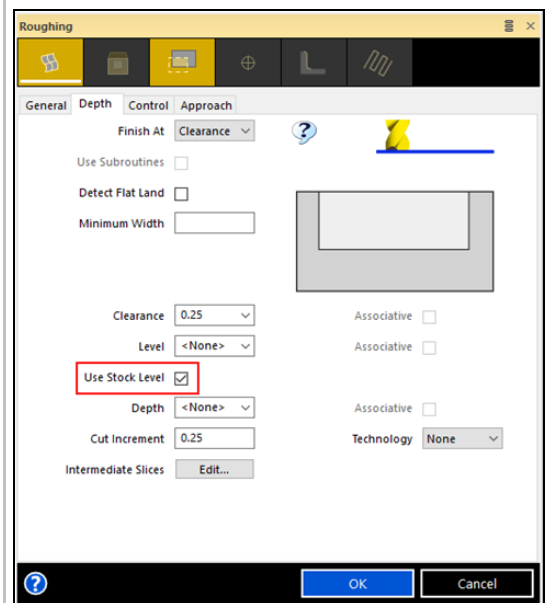
### Roughing Cycle - Use Stock Level

For this release, a new **Use Stock Level** field has been added to the Depth tab.

This is used in conjunction with Current Stock:

- Check to use the Current Stock Level.
- Uncheck to set the Level independently of Stock Level.

When **Use Stock Level** is checked and the **Depth** is picked, the **Depth** value is now set to an Associative Depth to ensure that the toolpath is generated at the correct height.



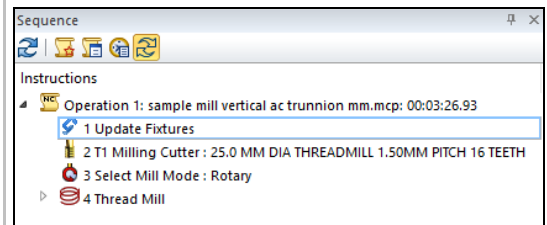
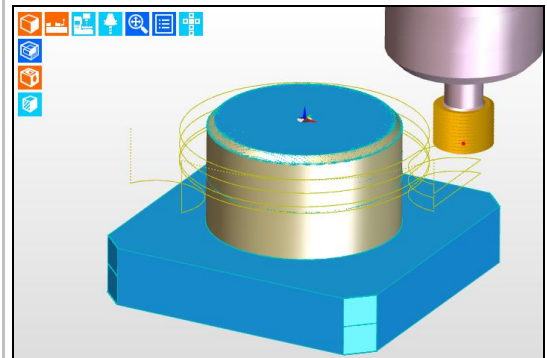
### Thread Mill in Rotary Axial

For this release, the Thread Mill cycle has been enhanced to allow it to work in rotary mode.

The user will now be able to create the cycle when rotary mode is enabled.

#### Notes:

1. Applies to Axial oriented machining only.
2. Applies to Mill, Turn and MTM environments.



```

X73.763 C15.73
Z5.0
G94 G1 Z-4.353 F500.0
G17 G3 X51.0 Z-4.5 I-2.763 J0.0 K0.588 C0.0 F375.0
G2 Z-6.0 I-51.0 J0.0 K1.5 F625.0
G3 X73.763 Z-6.147 I20.0 J0.0 K0.588 C-15.73 F375.0
G1 Z-10.353 F500.0
C15.73
G3 X51.0 Z-10.5 I-2.763 J0.0 K0.588 C0.0 F375.0
G2 Z-12.0 I-51.0 J0.0 K1.5 F625.0
G3 X73.763 Z-12.147 I20.0 J0.0 K0.588 C-15.73 F375.0
G1 Z-16.353 F500.0
C15.73
G3 X51.0 Z-16.5 I-2.763 J0.0 K0.588 C0.0 F375.0

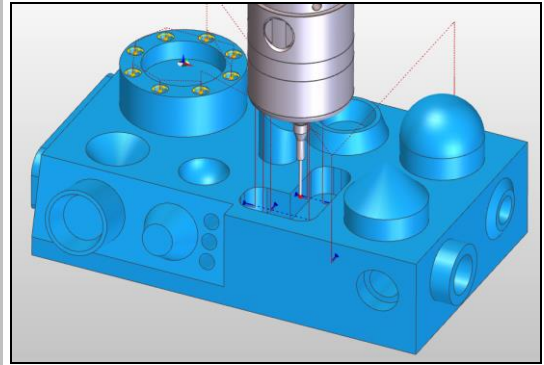
```



### Edgecam Inspection

Edgecam Inspection is a new product that performs inspection measurement on the machine tool either as a standalone system or as an add-on module to Edgecam Milling. The system allows you to probe the component with various measuring methods and generate the NC code for the machine. The results can be returned from the controller, analysed and compared with the model to generate an inspection report.

**Note:** To use this product you will need either an Edgecam Inspection System (ENINS-S) or Edgecam Inspection Module (ENINS-M).

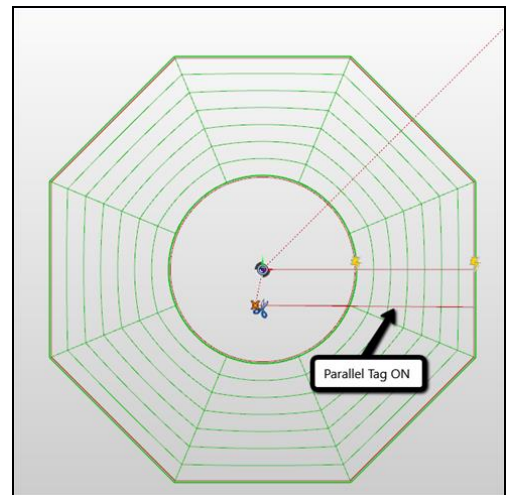
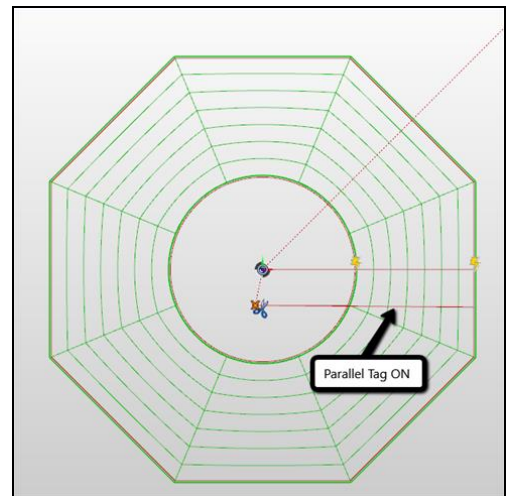
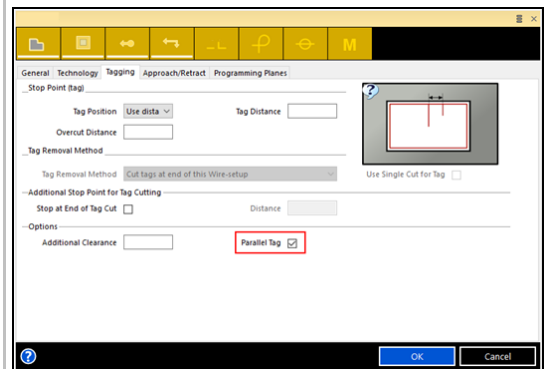


## Wire Enhancements

### 'Parallel Tag' option added to the Smart and 4 Axis cycles

A **Parallel Tag** option has been added to the **Tagging** tab for the 4 Axis and Smart cycles.

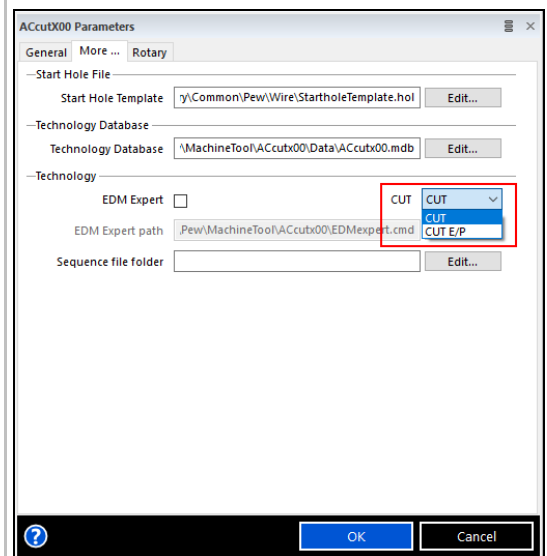
When checked, it will force the Tag to be parallel.



### Support EDM Expert Cut Type variations

In Machine Configuration, it is now possible to choose between **Cut** and **Cut E/P**:

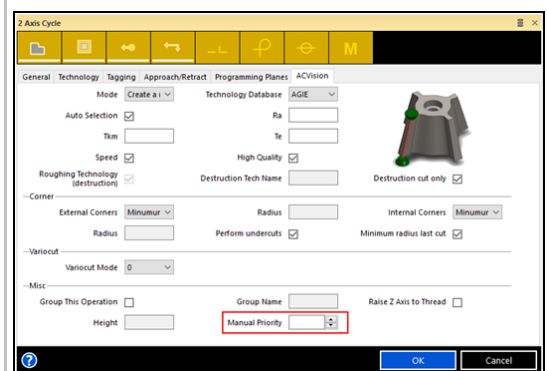
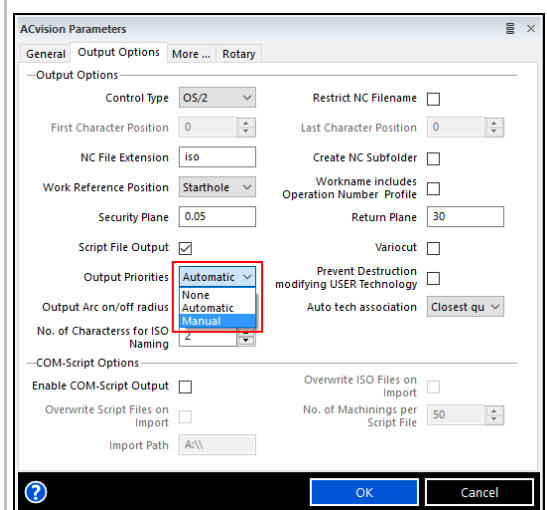
- When selecting **Cut E/P**, it is possible to load the 'Cut E350/E650' or 'Cut P350/P600' technology.
- When selecting **Cut**, it is possible to load the 'Cut 200/300' or 'Cut 400' technology.



### Manual option added to 'Output Priorities' (ACVision)

The **Output Priorities** option has changed from a checkbox to a drop-down list, allowing it to be **Off**, **Automatic** or **Manual**:

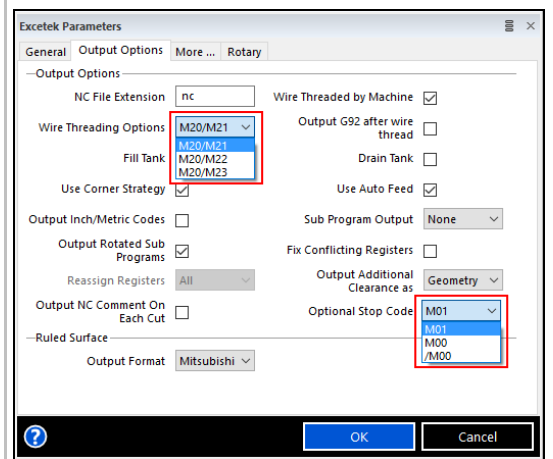
- The **Automatic** option will give the same output as in previous versions, following the order of instructions in the sequence.
- If using **Manual**, the priority will be manually set for each cycle.
- The **Manual Priority** option is available on the **ACVision** tab in the cycle.



## Excetek Output changes

In order to support other machine tools, the following options have been added to the **Output Options** tab of the Excetek post:

- **Wire Threading Options.**
- **Optional Stop Code.**

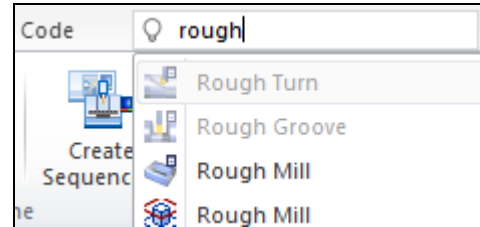
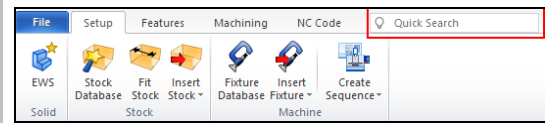


## Interface Enhancements

### Quick Search

A Quick Search function has been added to the Ribbon bar providing quick access to commands. This can be useful when the command is not on the ribbon, by default.

The function searches for any command names or tooltips containing the search string. Commands which are not available for the current license or not applicable to the current session, for example, Turning commands in a Milling session, will be greyed out.



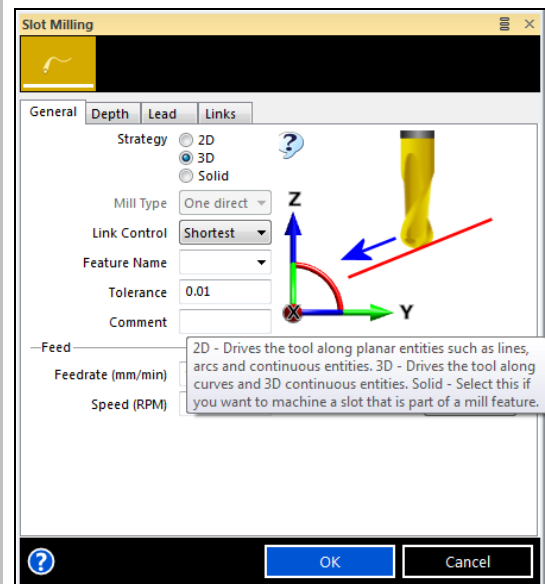
### Formulas in Dialogs

We are in the process of making every Edgecam modifier a potential JavaScript statement. This means that the **Cut Increment** modifier in Roughing, for example, can have a value as input, a reference to another field or even a command that will look up PCI variables, access your computer data, run a macro and then generate a value.

### Images on Dialogs (Milling Cycles)

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

- Slot Milling.
- Project Flow Curves.
- Project Circular Pattern.

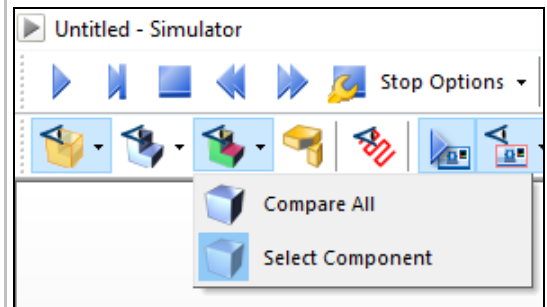


## Simulator Enhancements

### View Comparison for selected Components

For this release, we have introduced support for View Comparison on a selected component or on all components.

A drop-down list has been added to the **View Comparison** button containing the **Compare All** and **Select Component** options.

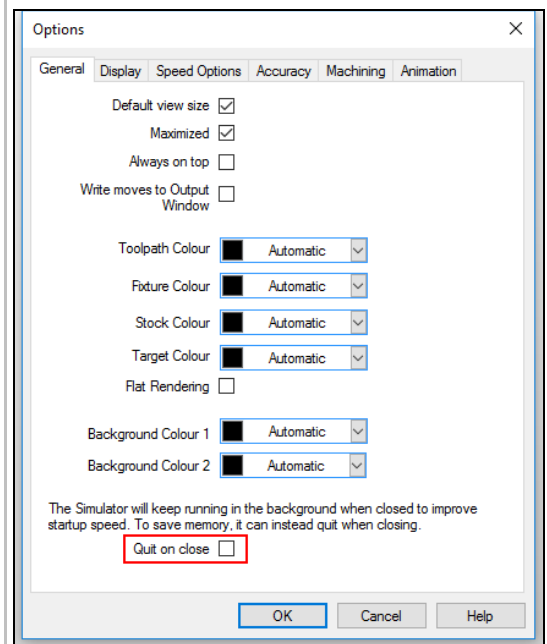


### 'Quit on close' on single occasion

Having the **Quit on close** option disabled in Simulator keeps the application running in the background and, therefore, enhances loading times for Simulator. Optimally, this option should always be disabled to maximise performance.

However, there are situations where Simulator needs to be fully closed. To allow this without forcing users to have **Quit on close** enabled all the time, the user can now hold down the **Ctrl** key while exiting the application to fully quit on a single occasion.

**Note:** This does not change the setting of the **Quit on close** option.

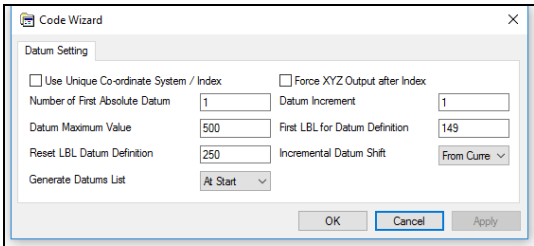


Code Wizard Enhancements

Datum Calculation for TNC Templates

For this release, four new parameters have been added to the **Datum Setting** tab which will enable you to configure datums in the same manner as subroutines:

- **Generate Datums List.**
- **First LBL for Datum Definition.**
- **Reset LBL for Datum Definition.**



## Post Processor Improvements

The Edgecam Post Processors have been improved in the following areas:

- **Lookahead macro improvements**  
Lookahead has been made more robust when subroutines are in use and the instruction pointers are more consistent. Out of range pointers will not cause undue problems and a safer approach is to check the value before use.
- **Probing Speed warnings are now bypassed**  
When Probing was added to turning, unnecessary warnings were being generated in some situations. Changes have now been made in the milling and turning Code Wizard templates to suppress speed warnings when a probe tool is loaded. The speed check is irrelevant because Edgecam forces a Spindle Stop (M5).
- **IndexG200 standardisation**  
The IndexG200 template has been aligned with the other templates resulting in changes to the source files but no changes in the output code. These changes are mainly related to tidying up the source code, rationalising variables and making the template use the common procedures that already exist. As a consequence, some options that were previously missing, for example, NC Style Tab, can now be added.
- **Adjustment for Helix arcs on TNC based machines**  
Helix arc centres have been adjusted to resolve issues with alarms being raised on TNC controls. An additional procedure has been added to the TNC milling templates, which is only called when the option for adjusting Helix arcs is set to ON in the NC Style tab. The arc is processed using the polar angle and increment. The error between the generated arc end point and the true arc end is then analysed, and, if too large, the arc is reprocessed with a shifted centre until the best option is determined. This should fix helical arcs on tapered threads, especially those for Waveform Roughing approach moves.
- **High Speed mode in MTMs Toolchange dialog**  
The High Speed mode in the Toolchange dialog was not available in MTM machines in previous releases. This has now been added and the Code Generator controlled modifier will work for all machines.
- **Tool ID Conversion (Integrex template)**  
The **String for Type O** Tool ID conversion parameter in the NC Style tab is now available.
- **POCKET4 order of tokens**  
Due to different terminology used by Siemens, **POCKET4** was previously configured with the wrong order in the Helical Hole Cycle. The order of the [CY\_RPLANE] and [CY\_ZDEPTH] tokens has changed.
- **B Axis Turn Implementation**  
The turning templates have been changed to support the new B Axis Turn cycle and include new tokens. You will need to upgrade to the latest templates to use the new cycle.
- **THREADCHAMFMCODE token added to missing templates**  
The THREADCHAMFMCODE token has been added to the Siemens and IndexG200 templates.
- **NEXTSPEED**  
In previous releases, NEXTSPEED had a limitation where a **Move To Toolchange** or **Move To Home** would cause the lookahead to stop. This has now been changed for the milling environment and NEXTSPEED will no longer stop on **Moves to Home/Toolchange**.



## Code Generator Enhancements

### Stock information on Post Processor

Stock sizes are now passed to Code Generator allowing the user to use that data to replicate those stocks on external simulators. Some applications also require billet information for other purposes.

Code Generator has a new %LOADSTOCK directive to load stock information.

#### Note:

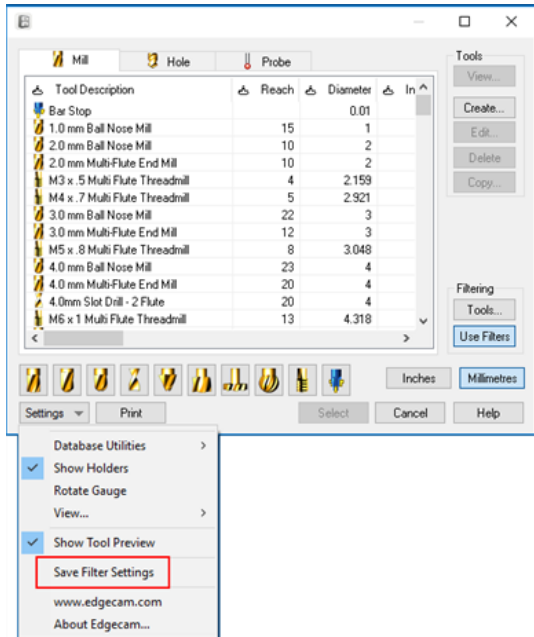
- There is no ordering of stocks, as there is no such concept in Edgecam. Code Generator will load the stocks in the order defined in internal state manager.
- There is no information about assignment of stock (i.e. main or sub-spindle).
- Stock information relies entirely on the stocks defined in the setups. If a stock is incorrectly defined, no information will be available in the output. Therefore, old parts may not work; geometry might need to be recreated.
- The old Stock/Fixture command in the Edgecam Geometry menu can still generate stocks and is particularly useful when there is no solid model. When creating a sequence containing only wireframe entities, the user needs to deliberately select the stock for it to be part of a component.

## ToolStore Enhancements

### Option added to Save ToolStore Filters

A **Save Filter Settings** option has been added to save ToolStore filters when selecting a tool from the ToolStore:

- With **Save Filter Settings** checked, the filter settings are retained from the previous session.
- When using the **ToolStore** button in Edgcam, any previously set filters such as Tool Type, Units, Use Filters and Filter Tool settings are retained for subsequent tool selections.
- When using the **Find** button from the Toolchange dialog, the Tool Type filter is set to the dialog tool type setting while retaining any previously set filters.
- When using ToolStore as a standalone application, the filter settings are retained from the previous ToolStore session.
- When **Save Filter Settings** is unchecked, filter settings will not be retained.

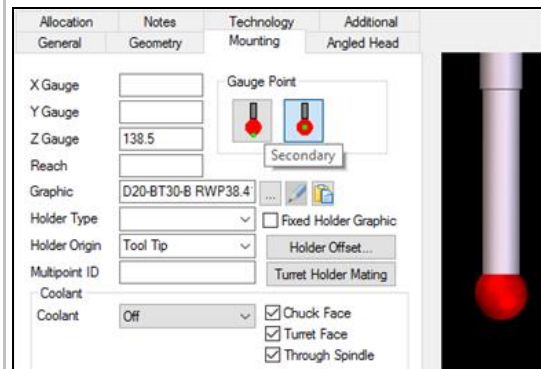


### Option added to set the Gauge Point at the centre of a Probe

An option has been added to set the **Gauge Point** at the centre of a probe.

This option is available on the Probe **Mounting** tab in ToolStore and on the Probe Toolchange **Loading** tab in Edgcam.

- The **Secondary Gauge Point** sets the gauge point to the centre of the probe stylus.
- Changing the **Gauge Point** between **Primary** and **Secondary** adjusts the **Z Gauge** value by the radius of the probe stylus.



## Live Job Reports Enhancements

### Live Job Reports

Live Job Reports has been converted to a desktop application which will improve reliability when working with LocalDB databases:

- As a desktop application, Live Job Reports is designed to be used by the logged in user on the computer.
- In previous versions, Live Job Reports was a windows service; this had reliability issues when running on LocalDB databases specifically when the user running Live Job Reports was different to the user that installed it.
- A Live Job Reports Windows Service is still installed (displayed on View Local Services) but this no longer runs the Live Job Reports site. It is still required to execute commands which demand higher privileges; specifically, open / close ports; manage firewall rules and to provide a static port used by remote searches.

**Note:** The User Interface has not changed.