

## We are pleased to release NCG CAM v20.0.07

This point release fixes a few issues that have been detected. There are other general ongoing enhancements too. Note v20.0.06 was not release as we found an issue after creating the installer.

### General

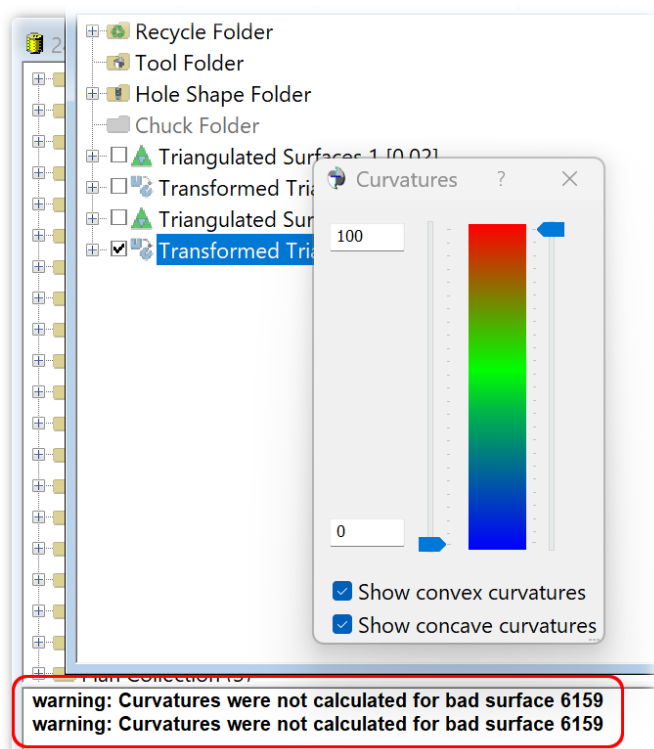
9127: Rest Finishing: A problem has been fixed where passes for some steep areas might not be generated.

9299: Tool Database: There was a problem when loading and restoring a tool database to the Tool Folder, causing holders to be lost, this has now been fixed.

9405: Rest Finishing: A problem has been fixed where badly formed passes, usually in steep areas could be generated.

9987: Selected Surface Waterline: A problem has been fixed where passes might not be generated for very narrow surfaces.

10041: Surface analysis: A problem has been fixed where the presence of a component surface that could not be triangulated would prevent the display of curvatures or draft angles for all component surfaces. Warning messages are now issued in the database window for surfaces for which analysis values could not be calculated.



10137: Stock models: A problem has been fixed where a difference between the XY and Z thicknesses on a cutter could lead to an inaccurate stock model.

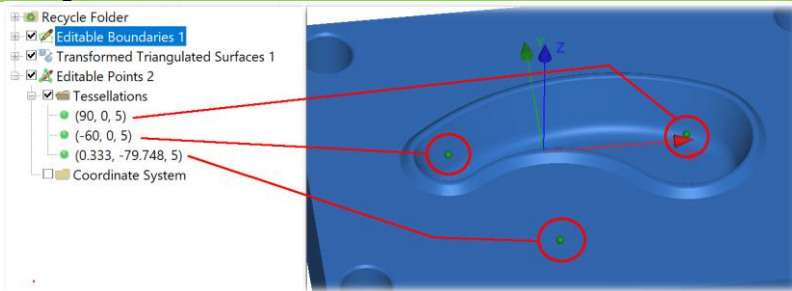
10553: Plans: Fixed an issue with deleted plans not being automatically cancelled. Plans no longer need to be paused to be automatically cancelled. This will also fix ticket 10673.

This would happen if a plan was being calculated and possibly had other dependent plans. Changing that plan or a plan it depended on, would calculate the original planes, then recalculate them for the changes. Now the original plans are cancelled, and only the new plans are recalculated.

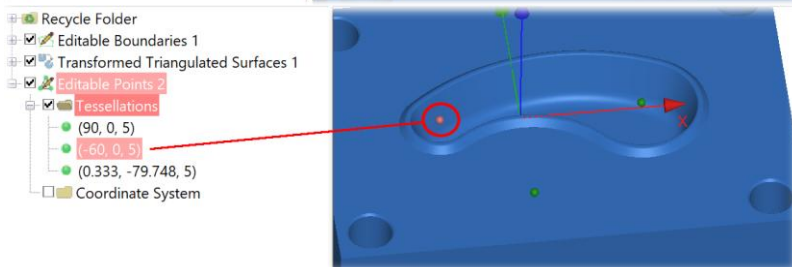
10572 and 10796: Five-Axis: Deburring - The default retract direction for 3+2 machining has been modified to retract along the tool axis and then Z, in the same way we do for other Five-Axis options.

10611: Coordinate System: It is now possible to select single points from points plans to perform and use "Set Coordinate System" to transform the selected item(s) rather than having to create datum points first. If datum points are selected the behavior will be as in previous versions.

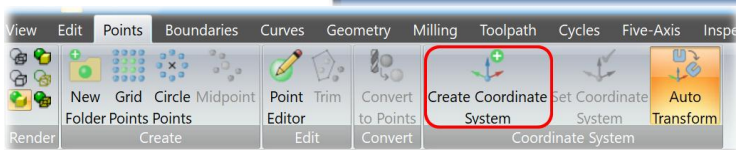
### Mini guide



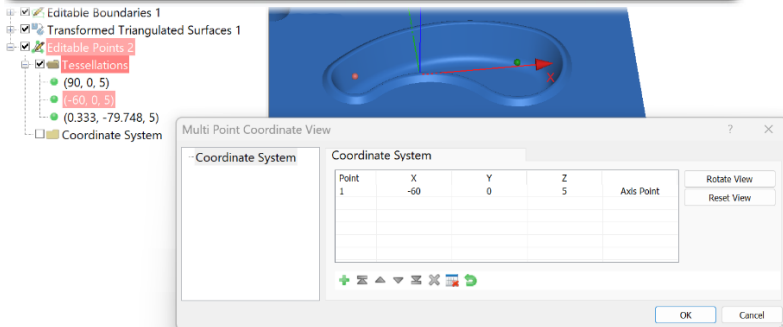
A points folder with one or more points



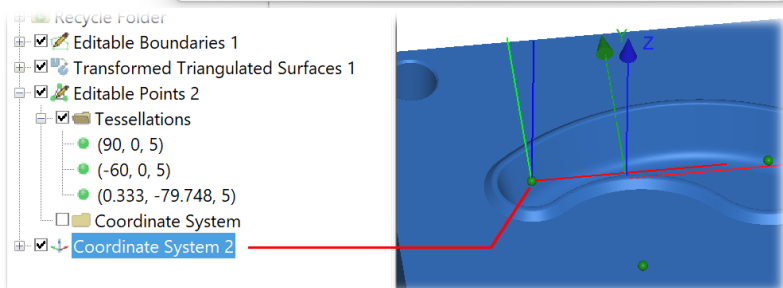
With the folder selected, also select the point to be used for the new coordinate system.



Pick Create Coordinate System in the Points ribbon.

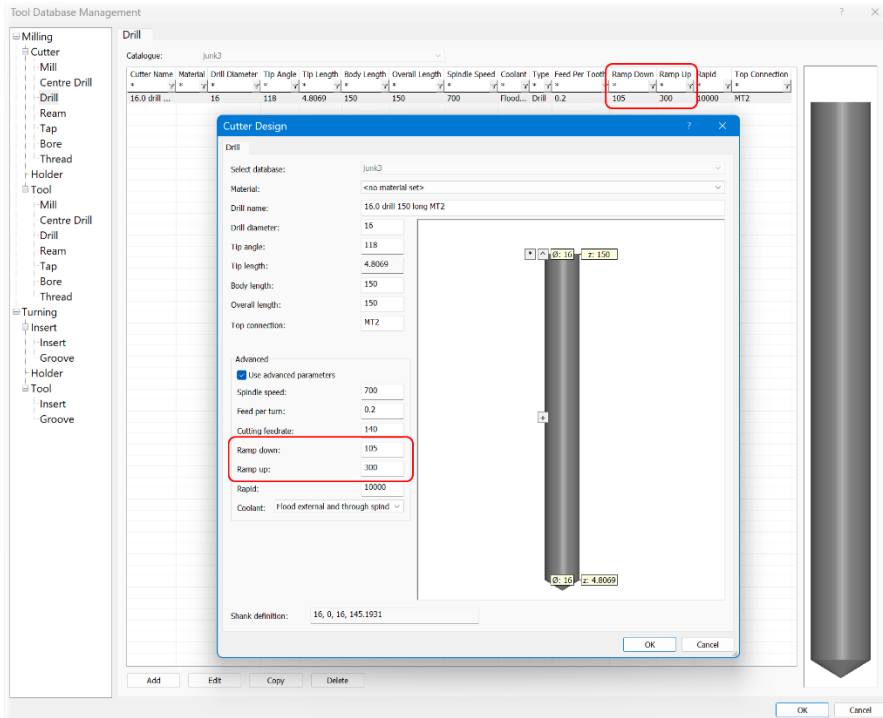


That will display the Coordinate View dialog, with the selected point loaded.

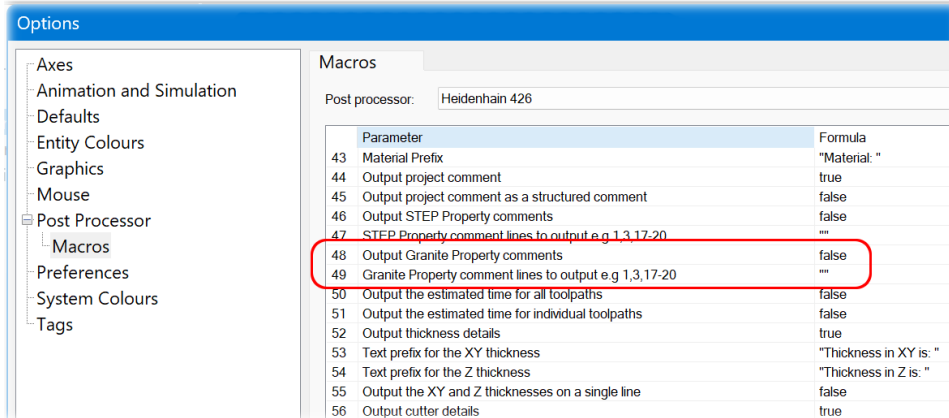


Clicking OK will create the Coordinate system point.

10651: Tool Database: The RampDown, RampUp and Rapid rates to cycle cutters have been added to the dialog to be consistent with the plan dialog tool pages. Some Heidenhain canned cycles use the Ramp-up feedrate to retract in a hole.



10665 and 10792: Import: Added support to store Creo / Granite file properties in the database, allowing them to be viewed again through File > Properties when the file is reloaded. Also added post-processing support for Creo component properties so selected properties can be output as comments in the tape file.



In the post, there is the option to output the Creo properties. Then the lines to be output can be set.

Creo File Properties

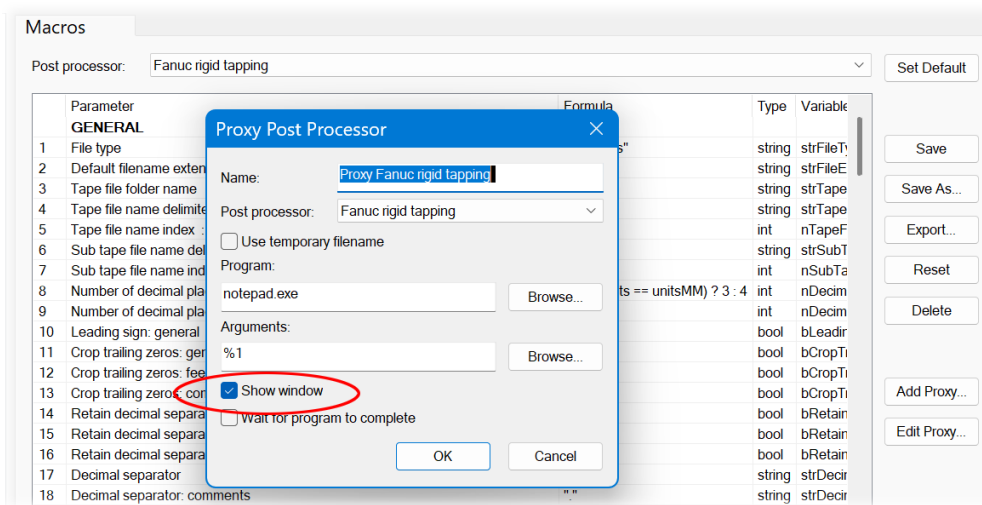
| Number | Name              | Value    |
|--------|-------------------|----------|
| 1      | KODA              | ...      |
| 2      | MATERIAL          | 1.2343   |
| 3      | KONSTRUIRAL       |          |
| 4      | NAZIV             |          |
| 5      | KOS               |          |
| 6      | COMMENT           |          |
| 7      | PTC_MATERIAL_NAME | GRAPHITE |
| 8      | STANDARD          |          |
| 9      | DOBAVITELJ        |          |
| 10     | MASA              | 0.53205  |
| 11     | OPOMBA            |          |
| 12     | PARTNAME          |          |
| 13     | SPECIAL           |          |
| 14     | DATUM             |          |

Picking Properties for the database and then picking the 'Properties' tab will display the names and values.

10681: ModuleWorks: Updated to the new MW2026-04 libraries for the five axis module.

10685: User Interface: Fixed an issue with the status bar where the plan progress display was not being updated unless the process manager dialog was open.

10686: Post-processor: Added a proxy post-processor option that controls whether the proxy process window is shown. When creating a Toolsheet, a command window is no longer briefly shown.



10687: Toolsheets: If the default viewer option for XML or CSV toolsheets is left blank (in Options > Defaults) then it will now default to using the Windows default viewer for HTML and CSV files respectively.

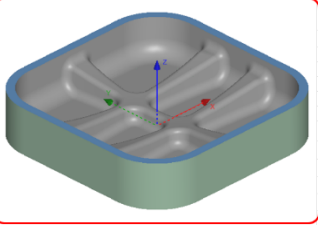
10704: Import: The Datakit libraries have been updated to 2026.2. Supports Parasolid V38.1. Also fixes ticket 10582.

About NCG CAM 20.0

| Module      | Version               | Licensed |
|-------------|-----------------------|----------|
| Catia v4 3D | Releases 4.15 to 4.24 | Yes      |
| Catia v5    | Releases 7 to 33      | Yes      |
| GRANITE     | 18.0.0.0              | Yes      |
| Parasolid   | Parasolid V38.1       | Yes      |
| Rhino       | Rhino V8              | Yes      |
| SolidWorks  | SolidWorks 2026       | Yes      |

10708: Post Processor: Hurco- Fix Drilling dwell prefix. The default had incorrectly been set as "G04", when it is more likely to be "P". This should not affect any existing post as the correct prefix will have been defined in the post processor setting.

10714: Editable Toolsheets: Fixed a problem with the Editable Toolsheet Builder failing to add images to the toolsheet.

| Sample Operations & Tool Setup Sheet                                              |                                         |                  |                                 |          |          |             |               |                                                                                     |         |           |
|-----------------------------------------------------------------------------------|-----------------------------------------|------------------|---------------------------------|----------|----------|-------------|---------------|-------------------------------------------------------------------------------------|---------|-----------|
|  |                                         | Project Engineer |                                 |          |          |             |               |                                                                                     |         |           |
|                                                                                   |                                         | Date             | 10-Jun-26                       |          |          |             |               |                                                                                     |         |           |
|                                                                                   |                                         | Project Database | D:\NCGCAM_Testing\test_post.dca |          |          |             |               |                                                                                     |         |           |
|                                                                                   |                                         | Project Folder   | D:\NCGCAM_Testing\              |          |          |             |               |                                                                                     |         |           |
|                                                                                   |                                         | Post Processor   | Mikron HSM 400U 5 axis ISO Demo |          |          |             |               |                                                                                     |         |           |
| Project Comments                                                                  |                                         | NCG Version      | 20.0.06                         |          |          |             |               |                                                                                     |         |           |
|                                                                                   |                                         |                  | Tool Path Limits                |          | X        | Y           | Z             |  |         |           |
|                                                                                   |                                         |                  | Minimum                         |          | -86.5    | -75         | -4.995        |                                                                                     |         |           |
|                                                                                   |                                         |                  | Maximum                         |          | 94       | 75          | 28.981        |                                                                                     |         |           |
|                                                                                   |                                         |                  | Part Limits                     |          |          |             |               |                                                                                     |         |           |
|                                                                                   |                                         |                  | Minimum                         |          | -65      | -65         | -5            |                                                                                     |         |           |
|                                                                                   |                                         |                  | Maximum                         |          | 65       | 65          | 23            |                                                                                     |         |           |
| Op. No.                                                                           | Operation Name                          | Tool No.         | Tool Description                | Diameter | Corn.Rad | Speed (rpm) | Feed (mm/min) | Thick XY                                                                            | Thick Z | Est. Time |
| 1                                                                                 | Waterline Toolpath 1 [20x0, 0]          | 1                | End Mill[20x0 60, 0]            | 20       | 0        | 2000        | 1194          | 0                                                                                   | 0       | 00:02:08  |
| 2                                                                                 | Boundary Toolpath 3 [20x0, 0]           | 1                | End Mill[20x0 60, 0]            | 20       | 0        | 2000        | 1194          | 0                                                                                   | 0       | 00:00:22  |
| 3                                                                                 | Five-Axis Boundary Toolpath 2 [12x6, 0] | 2                | Ball Nose[12x6 50, 0]           | 12       | 6        | 4000        | 1200          | 0                                                                                   | 0       | 00:00:22  |

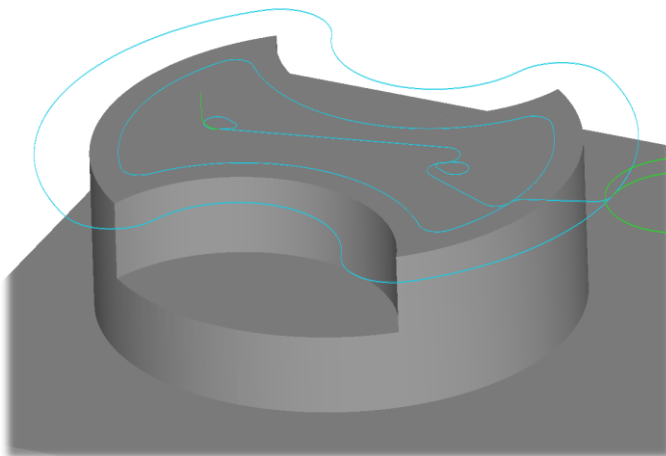
10721: Import: There was a problem with missing STEP file property values when loading, this has now been fixed.

10722: Turning: There was an issue where the tool tip was sometimes calculated incorrectly when the lead angle was less than 90 degrees, this has now been fixed.

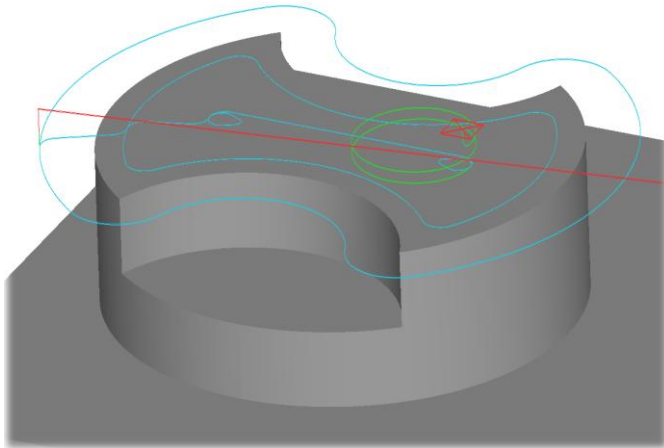
10727: Boundary Passes: There was a problem when doing True Surface Machining could in some cases cause NCGCAM to crash, this has now been fixed.

10729: Theoretical Rest Areas: The overall robustness and quality of Theoretical Rest Area plans has been improved.

10731: Linking: Horizontal Area Passes. For instances where the Outside to Inside option is enabled, a restriction has been removed which could prevent the creation of on-surface s-shaped moves within local pockets.



The new linking gives the toolpath shown to the left.  
The linking is from the outside and the linking moves are on the surface.



The old linking gave the toolpath shown to the left.

There is a helical ramp into the part..

10733: Post processors: Heidenhain, ISO, ISO Turning, Siemens, and Datron NEXT: Comments: Added an Engineer prefix, so rather than just outputting the name to the NC file, it is possible to output Engineer: <name> or Programmed by: <name>. The default is not set so existing posts processors are not affected. Also added the option Output material and a Material prefix, the Output material is false by default again so that existing posts processors are not affected.

Macros

Post processor: Fanuc rigid tapping

| Parameter                                              | Formula      | Type   | Variable |
|--------------------------------------------------------|--------------|--------|----------|
| 45 Output DCA file folder path                         | true         | bool   | bOutput  |
| 46 Substitute the \ in the path for /                  | false        | bool   | bSubsti  |
| 47 Output engineer's name                              | true         | bool   | bOutput  |
| 48 Engineers Prefix                                    | ""           | string | strEngir |
| 49 Output Material                                     | false        | bool   | bOutput  |
| 50 Material Prefix                                     | "Material: " | string | strMate  |
| 51 Output project comment                              | true         | bool   | bOutput  |
| 52 Output STEP Property comments                       | false        | bool   | bOutput  |
| 53 STEP Property comment lines to output e.g 1,3,17-20 | ""           | string | strOutpu |

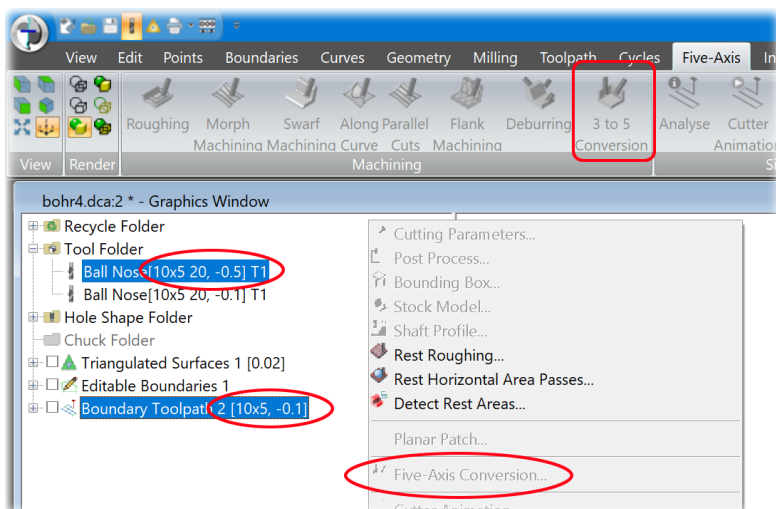
Buttons: Set Default, Save, Save As..., Export..., Reset

10738: Toolsheets: Added the project settings Material to the toolsheet.

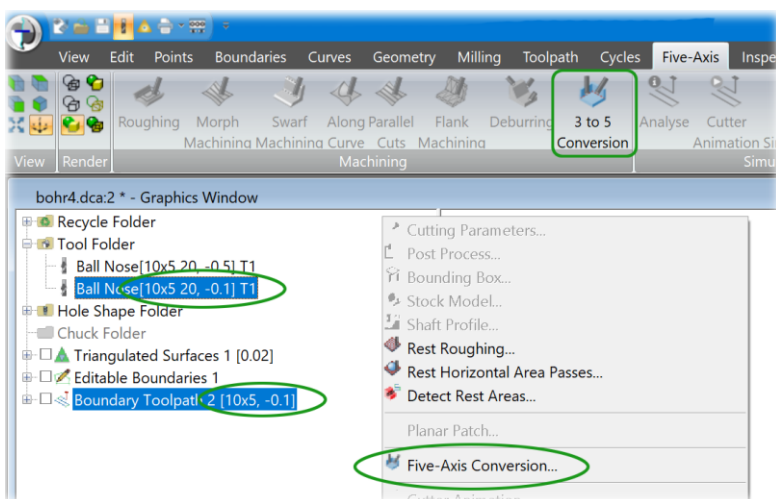
| NCG CAM                |                                                       | Date           |          | 26 June 2026 |           |               |          |       |         |
|------------------------|-------------------------------------------------------|----------------|----------|--------------|-----------|---------------|----------|-------|---------|
| NCG CAM Solutions Ltd. |                                                       | Post Processor |          | Roeders RZL  |           |               |          |       |         |
| Database name          | D:\Jobs_Overseas\Germany\2026\2026_06_23_Post_Rodets\ |                |          |              |           |               |          |       |         |
| Project prefix         | Test_1.dca                                            |                |          |              |           |               |          |       |         |
| Project engineer       | development                                           |                |          |              |           |               |          |       |         |
| Project directory      | D:\Jobs_Overseas\Germany\2026\2026_06_23_Post_Rodets\ |                |          |              |           |               |          |       |         |
| Project material       | Copper                                                |                |          |              |           |               |          |       |         |
| Project comment        |                                                       |                |          |              |           |               |          |       |         |
|                        | Min                                                   | Max            | Min      | Max          | Min       | Max           |          |       |         |
| Total part size        | X                                                     | -50            | 50       | Y            | -20       | 20            | Z        | 0     | 50      |
| Motion limits          | X                                                     | -60.503        | 68.675   | Y            | -30.506   | 30.507        | Z        | 0     | 55.475  |
| Op No.                 | Name                                                  | Strategy       | Tool No. | Tool         | Thickness | Spindle speed | Feedrate | Rapid | Cutting |
|                        | Waterline Toolpath 3                                  | Waterline      | 4        | 20x          | 0.5       | 2200          | 10000    | 1200  |         |

10740: Geometry: The Z-Trim option was not always working correctly when a rotation is specified, this has now been fixed.

10742: Five-Axis: If selecting a tool from the Tool folder for the Five-Axis Conversion which has different thicknesses to the original passes will result in the Five-Axis Conversion being blocked.



While the cutters are the same size, the thicknesses are different. This is not permitted as the resulting toolpath may not be as expected because of the conflicting thicknesses.



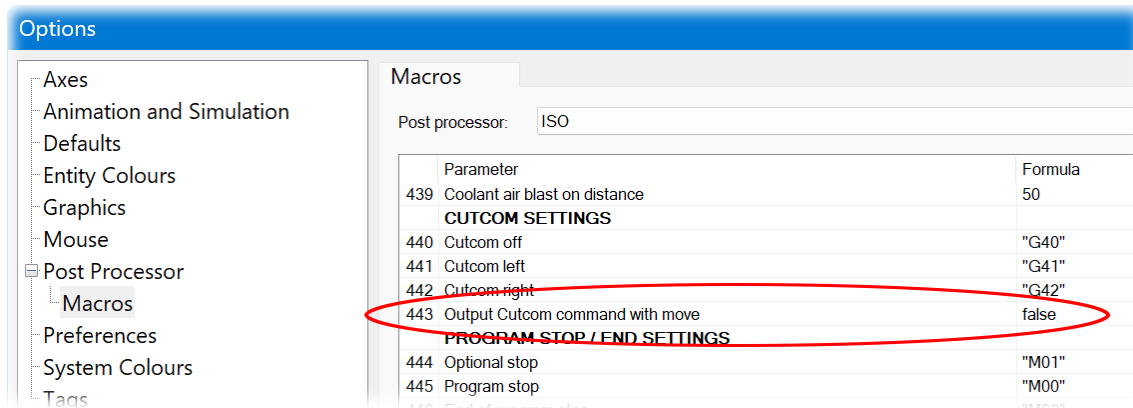
When the cutter and thickness details match, the Five axis conversion is available.

10749: Cutter Simulation: The toolpath name in the dialog was not always correct if the name had been edited, this has now been fixed.

10752: Tool Database: Exporting tools with composite holders to a text file would not read back in correctly, this has now been fixed.

10757: Tool Database: When exporting holders from the 'Tool folder' to a tool database, checks are now made to ensure they have a name, so not to cause problems with the tool database later.

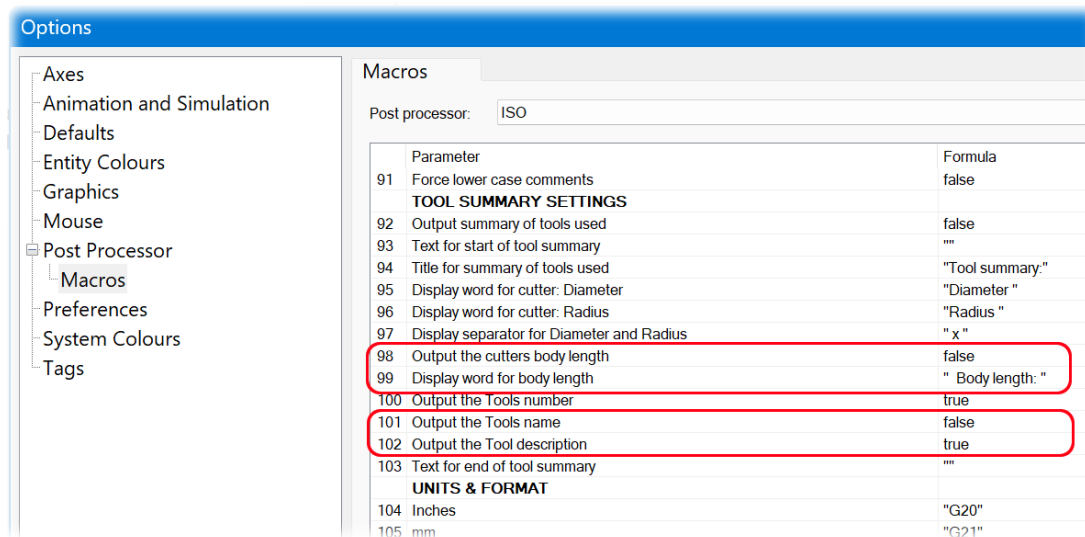
10758: Post Processors: ISO. Siemens and Heidenhain: Added the option "Output Cutcom command with move" to allow the cutter-compensation command to be output as part of a motion block. The default is false, so existing posts are not affected. Post processor documentation updated to match.



10759: Post Processors: ISO. Siemens and Heidenhain: Added some additional options for the tool summary tool details. "Output the cutters body length" allows the tools body length to be output, "Display word for body length" allows for text to prefix the body length, "Output the Tools number" allows the tool number to be output (this was always output before).

The "Output the Tools name", will allow the tool name to be output, and "Output the Tool description", allows the tool description to be output (this was always output before).

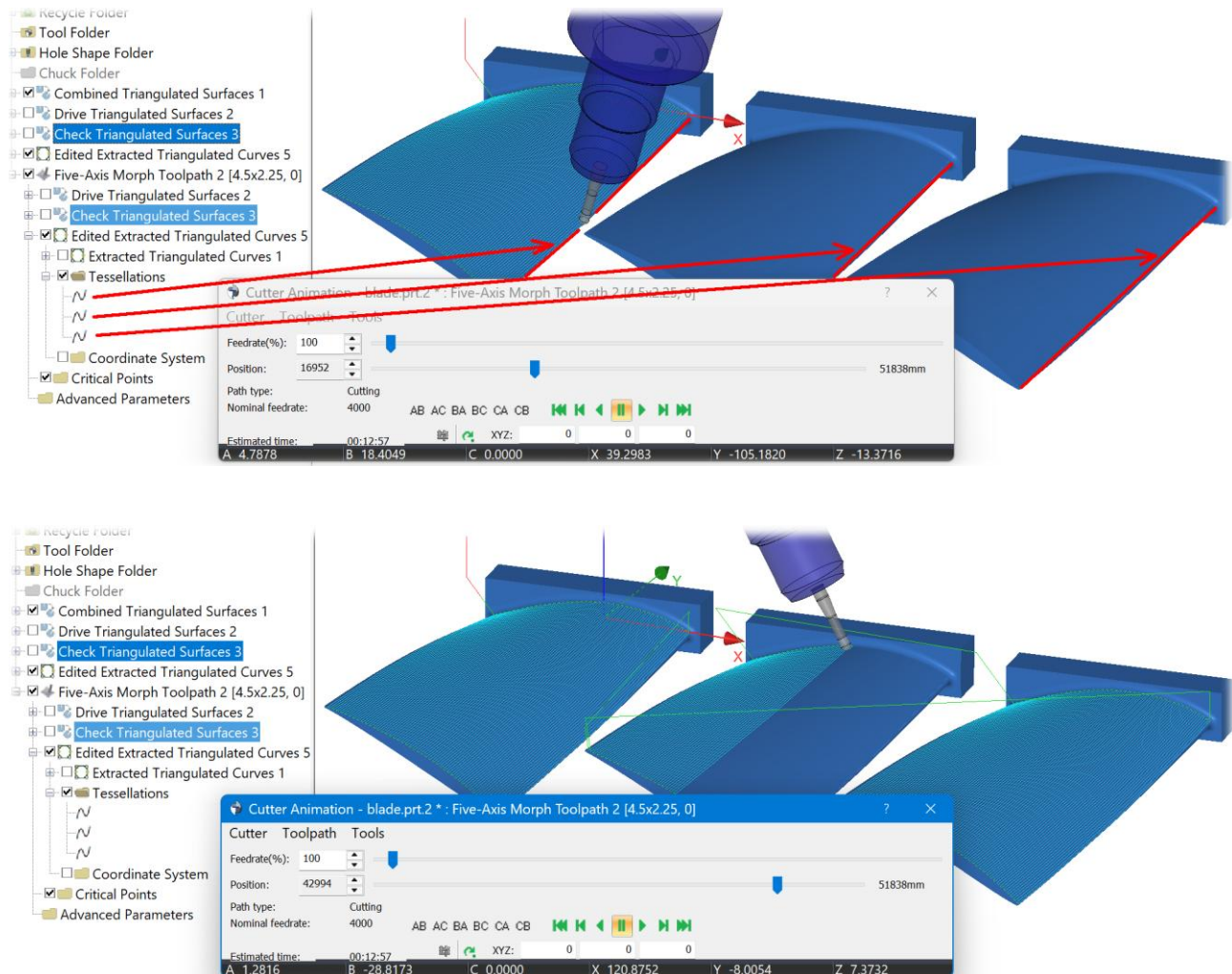
Note the Heidenhain already outputs the tool name or tool number (so does not have the "Output the Tools number" or "Output the Tools name" options in the same manner. Documentation updated to match.



10762: Database: Fixed a case where a database would not be saved correctly.

10770: An exception could be thrown when transforming an empty curves folder. This has been fixed.

10773: Five-Axis: It is now possible to select multiple drive curves for the "Morph to curve(s)" option, it had previously been restricted to a single curve. This allows multiple areas to be machined in one toolpath, previously it would have required a toolpath for each individual area. Each area must have its own single curve.



In the example above, there are three curves in the curves folder, the three areas (in the same Drive surfaces folder) can be machined as a single toolpath (in v20.0.05 and earlier that would have required three toolpaths – one if each area).

10774: Turning: Fixed a profile pass generation issue when the inserts tip radius is zero.

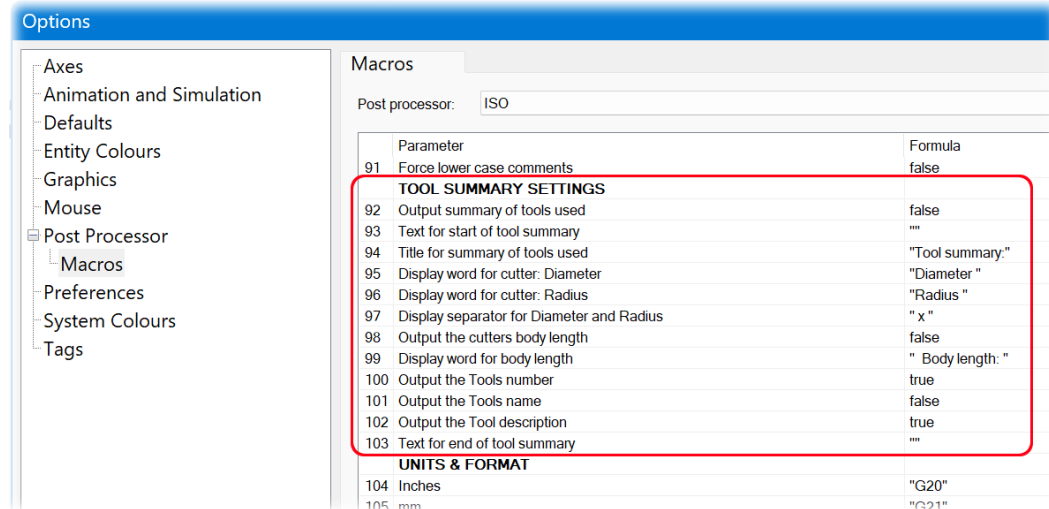
10780: User Interface: The control box for the Coolant type in tool database and plan dialog's has been extended slightly as it was not long enough.

10783: Export: Fixed an issue with the CSV export operations not working correctly.

10787: Cutter Animation: Swapping between the database and graphics windows while animating a toolpath was not restoring the Cutter Animation dialog correctly, this has now been fixed.

10790: Export: It was possible to overwrite an existing file when exporting a file for Detect Holes and Shaft Profile without knowing, this has now been fixed.

10791: Post Processors: ISO, Heidenhain and Siemens: The Tool Change Settings have had the Tool summary options moved to their own Tool Summary Settings, which is now after the Standard Start and End settings. This has no effect on any post processor output. Help files updated to match



## We are pleased to release NCG CAM v20.0.05 and 20.0.04

Version 20.0.04 was not released due to an issue with the installer. This issue was resolved in version 20.0.05. The software's functionality is identical in both versions.

In addition to bug fixes, this version also includes other functional improvements. The most significant new feature is the ability to view toolsheets in modern web browsers such as Chrome and Microsoft Edge.

**Note:** Due to the change to the toolsheets, there is currently an issue with the display of images in the Editable Toolsheet Builder (Excel toolsheet). If you use these toolsheets and rely on the images, we recommend that you do not install this version for the time being.

This issue will be resolved in the next version.

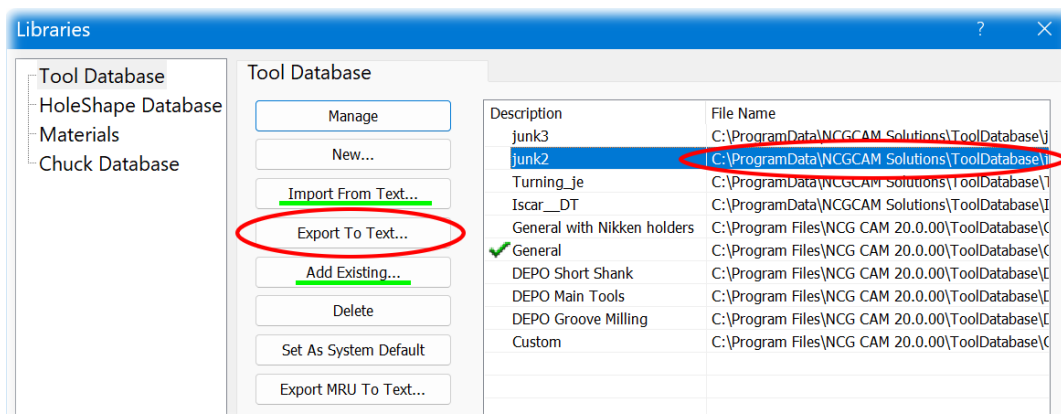
Tool databases opened in v20.0.05 are no longer compatible with older NCG CAM versions. For safety, we recommend creating copies of your tool library files (\*.tdb) before you open them in 20.0.05.

You can make a safe copy of your tool database in one of two ways:

- 1)
  - 1.1) From within NCG CAM, pick, **Application button > Libraries > Tool Database**.
  - 1.2) Select your Tool Database (single click).
  - 1.3) Pick: **Export To Text...**
  - 1.4) Save the <Tool\_database\_name>.txt to a suitable location.
  - 1.5) If you have more than one Tool Database, repeat 1.2 to 1.4 until all are exported.

You can then use the **Import From Text...** it reinstates the tool database if needed.
- 2)
  - 2.1) From within NCG CAM, pick, **Application button > Libraries > Tool Database**.
  - 2.2) Look for your tool database, and note the File Name, it shows the path and file name.
  - 2.3) Open a Windows File Explorer window, and browse to that location.
  - 2.4) Copy that file to a new location (or you could have a copy in the same folder, just change the Tool\_database\_name>.tdb(copy) to Tool\_database\_name>.tdb.save or Tool\_database\_name>.tdb\_<date>).
  - 2.5) If you have more than one Tool Database, repeat 2.2 to 2.4 until all are copied.

You can then reinstate the tool database if needed by using the **Add Existing...** option.



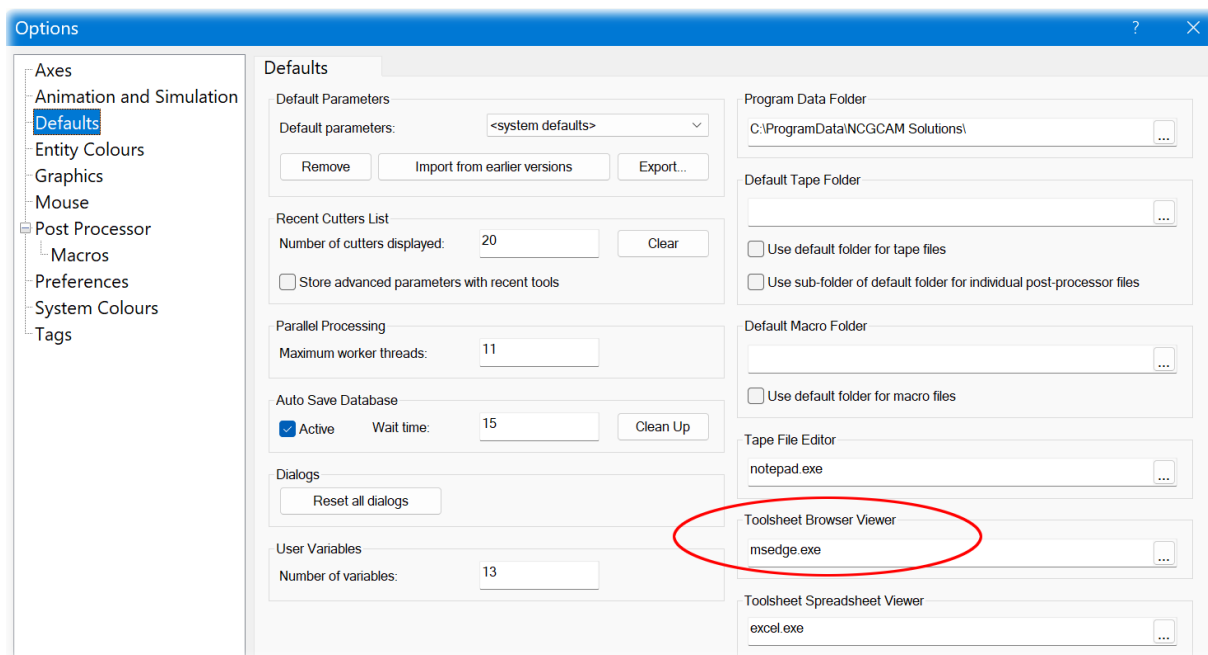
**6213 and 10659: Post processor: ISO-Siemens and ISO:** Additional HSM options have been added, the new options default to false so existing post processors are not affected.

The **Output the passes tolerance**, if set true, will output the passes tolerance after the HSM mode. If set true, the **Output the machining quality** will output a machining quality (Finishing or Roughing). The switch Finishing and Roughing is controlled by the toolpaths thickness, and the **If the thickness is less than this, use Finishing quality** option. The **Roughing quality setting** and **Finishing quality setting** are used for the machining quality number. Siemens say: 1 = Finishing, and 3 = Roughing. There is also a **Start HSM mode suffix** to allow for any additional parameter that may be required.

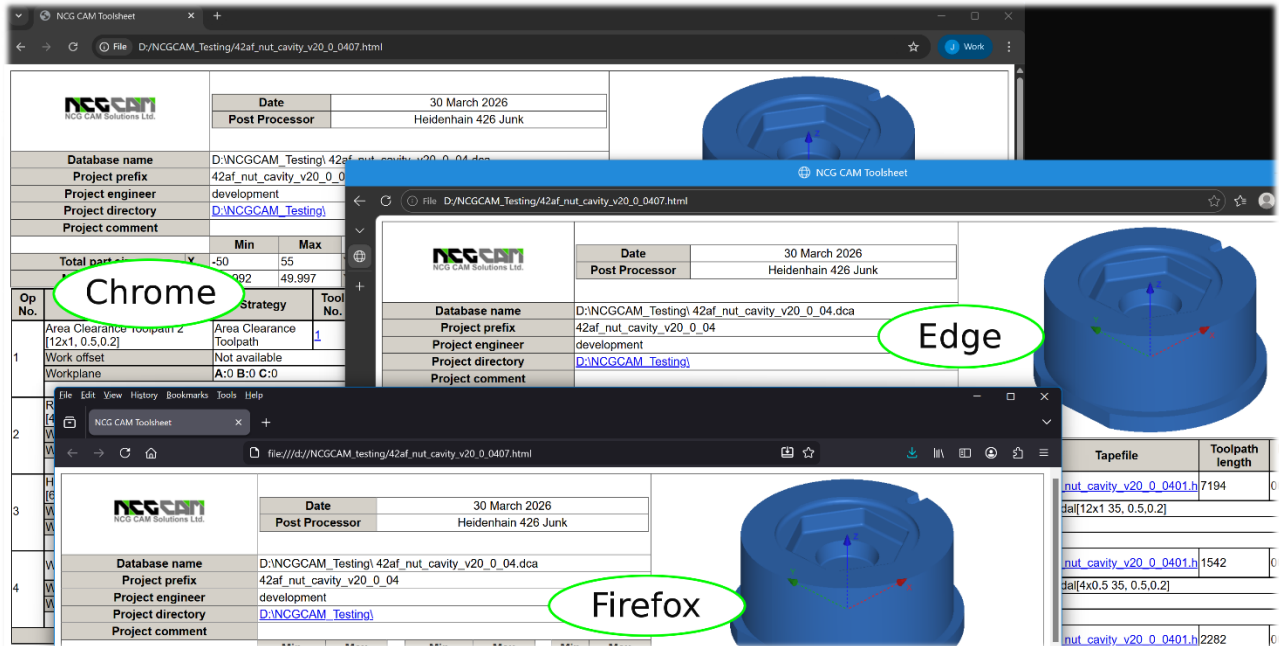
**6754: Tool database and Drills:** There is some preparation work done on the tool database structure. This means a tool database used in v20.0.04 will not then be useable in v20.0.03 or earlier. If you use the tool databases, making a safe copy may be wise (it also good backup practice to keep a safe copy of things like you tool databases, compound drilling cycles, post processors).

**9558: Toolsheet:** An HTML version of XML standard toolsheets is now automatically generated, which can be viewed via the context menu.

**Note:** That this change is not backwards compatible, installing 20.0.04 will cause toolsheet generation in previous versions to fail, the simplest solution is remaking the toolsheet in v20.0.04. The Toolsheet Post-Processor help has been updated. You will see a CMD window pop-up and then disappear as the toolsheet is created.

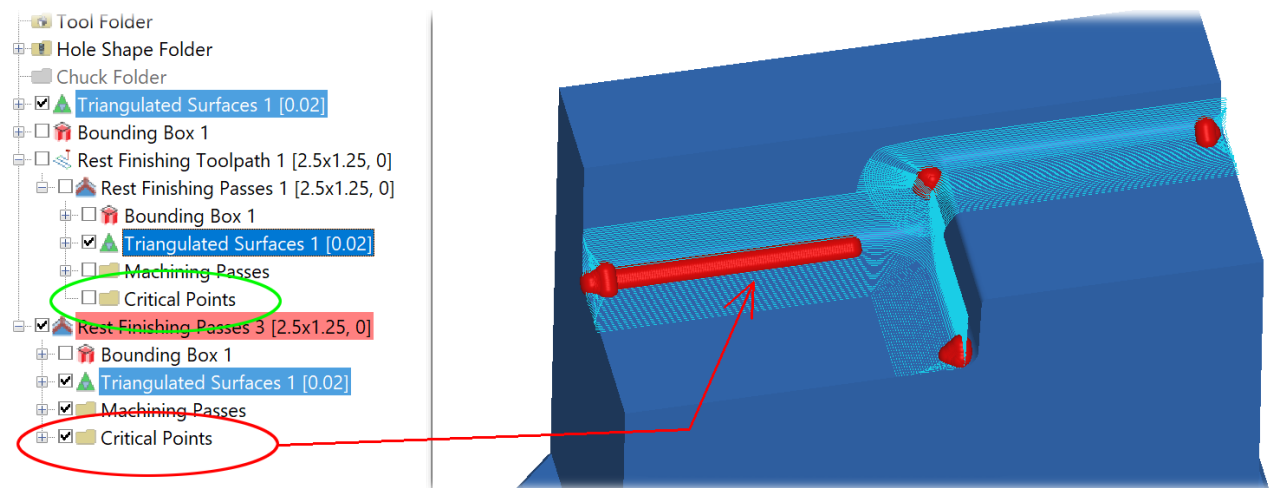


You may need to modify the path for your **Toolsheet Browser Viewer**. Just have firefox.exe or msedge.exe or chrome.exe, you will have msedge as part of the Windows operating system, but Chrome and Firefox would be browser's you have chosen to install.



The same toolsheet displayed in Chrome, Firefox, and Edge.

**9834: Rest finishing:** A Critical Points folder is now created under the Rest finishing passes folder. When the cutter has gone to the user specified maximum depth of cut without contacting any surface, the Critical Points folder is populated with points near the areas where material remains. Also, the plan folder is highlighted in light red (Warning) and the messages are output to the Database window.

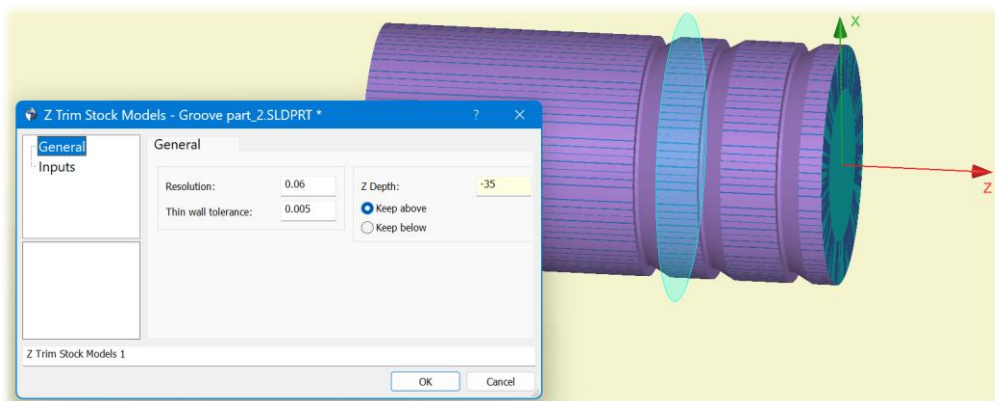


When the passes are created to depth, the Critical Points folder is empty (highlighted green), if NCG CAM can't machine to depth, the positions are written to the Critical Points folder (highlighted red) and the points are also displayed to make it clear where there is still material remaining.

**10180: Database:** Improved performance when saving, fixed crashes that could occur when attempting to write to folders without suitable 'write' permissions.

**10336: Stock Model:** The creation of closed prismatic stock models has been improved to be more accurate and contain fewer triangles.

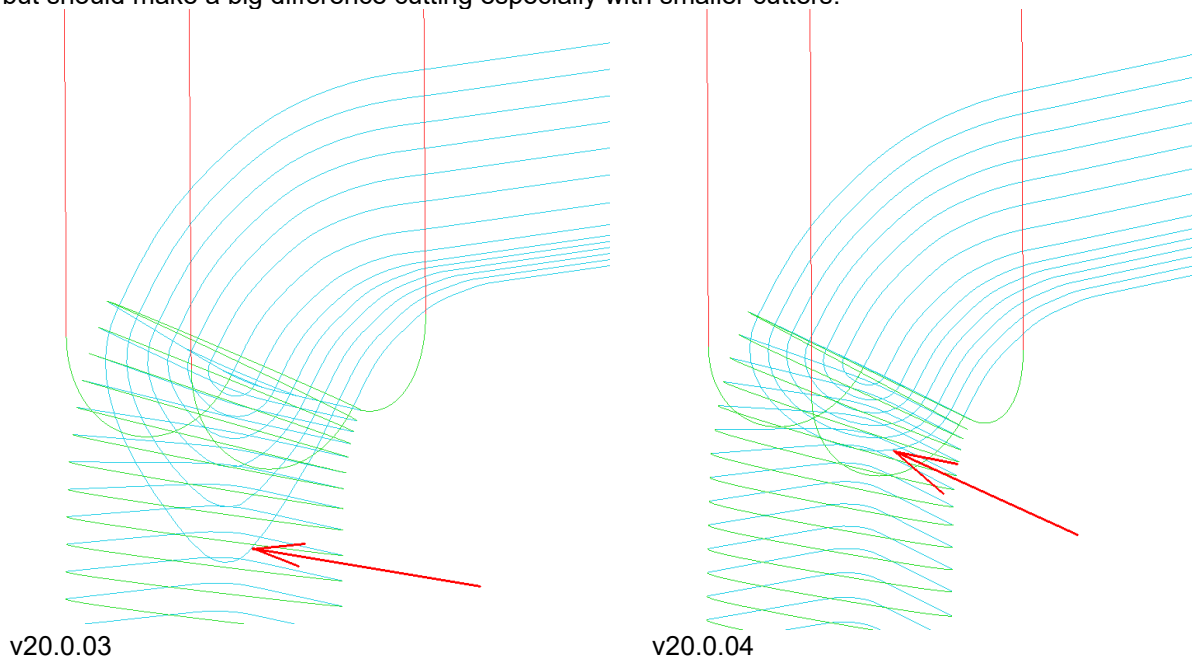
**10442: Stock Models:** The ZTrim stock model option for Turning had the milling Z Axis as the default Z axis, this has now been changed to the Turning Z Axis.



**10550: User Interface:** Fixed a problem with closed dialogs sometimes appearing momentarily when hovering over the program icon on the taskbar. Also, the 2D and 3D tool guide dialogs can now be closed by clicking on their ribbon buttons.

**10577: Rest Finishing:** Shallow area / spiral end connections that could include heavy cuts are now off-surface moves.

The same part and toolpath in v20.0.03 and v20.0.04 – The arrow points to the difference, visually small but should make a big difference cutting especially with smaller cutters.



**10616: Installer:** The DNCManager has now been removed from the installer, it is still available on request.

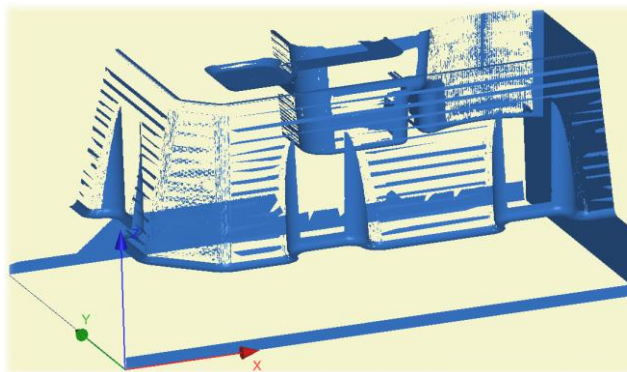
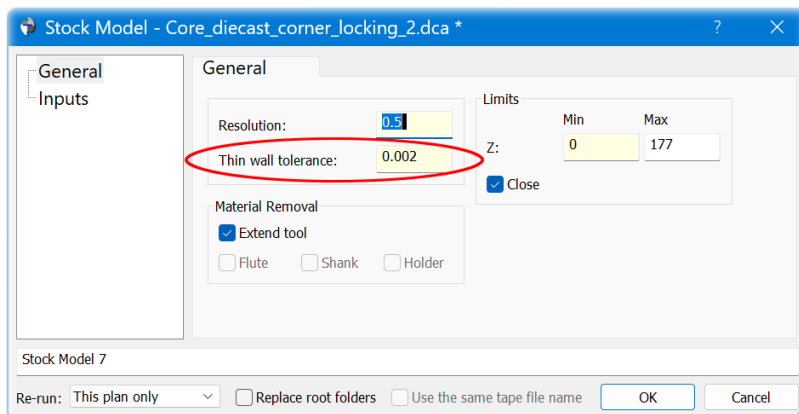
**10623: Linking:** Fixed an issue that could occur in some cases with profile ramps on looped fragments that also contained small stepdown.

**10626: Post Processing:** The toolpath page was not checking cycle tools correctly for duplicates, this has now been fixed.

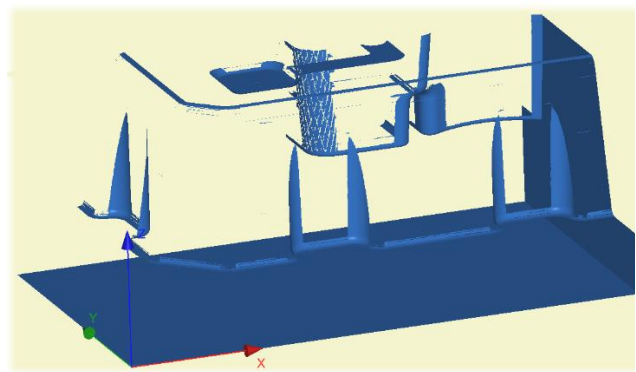
**10627: Post processor: ISO:** It is now possible to use the Tool Length as a Length / Height offset register position (this was previously possible, but it came from the tool number and still can). Added the options: Crop trailing zeros if outputting the height offset value, Use the height offset number for the diameter offset. Both are set false by default so existing posts are not affected.

**10628: User Interface:** The validation messages for angle constraints when defining a Turning Tool have been improved.

**10629: Stock Model:** The Stock Model plan dialogs now have the Thin Wall Tolerance parameter (for closed stock models), so the user can control the appearance of thin-walled sections in a stock model. The documentation has also been updated for this parameter. This can help achieve better results when subtracting stock models.



The result in v20.0.03 after subtracting one stock model from the other.



The result in v20.0.04 after subtracting one stock model from the other.

**10631: User Interface:** When transforming multiple plans, the transformed plans are now added to the tree in the same order as they were selected.

**10636: Drilling: Gouge protection:** The gouge allowance was being ignored for milling style drills (Bore, Thread, tap, Ream). This has been fixed.

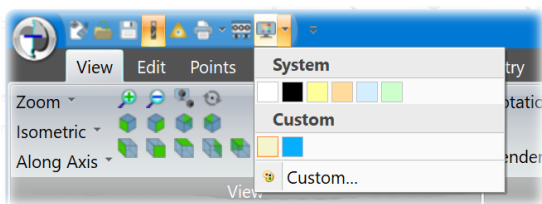
**10641: User Interface:** Added validation to ensure turning holder clearance does not exceed the insert tip radius.

**10643: Post processor: Siemens:** Added the option **Output the feedrate before the cycle** to allow the cycles feedrate to be output in the block before the cycle definition. This is set false by default so existing posts are not affected. If set true, the feedrate is not output with the first hole position in the cycle.

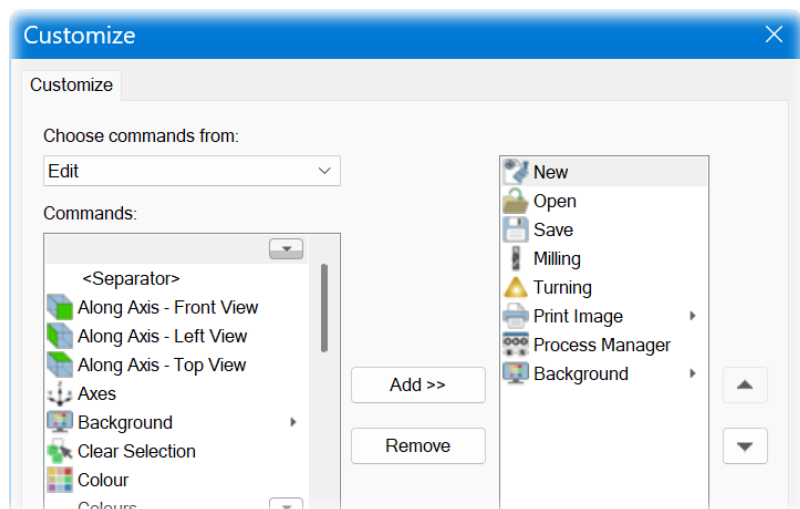
**10644: Post processors: Heidenhain:** Added the option **Output the TOOLCALL if spindle speed changes**. This is used if the spindle speed is reduced when helixing in for an adaptive clearance toolpath.

**10647: Adaptive Clearance:** If the Z limits are modified in the dialog the passes were not always calculated correctly, this has now been fixed.

**10655: User Interface:** If the Background colour command is added to the Quick Access Toolbar, the software could crash when the button is used when re-opening a dca, this has now been fixed.



The background button is not in the Quick Access Toolbar by default. It and others can be added via the **Custom...** option.

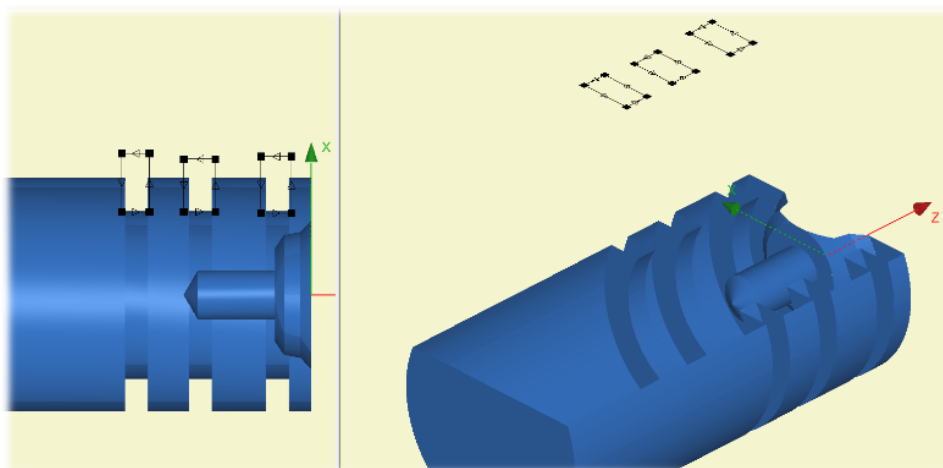


**10658: Import Tool:** There was a problem importing a tool from a text file created in 20.0.02 into 20.0.03, this has now been fixed.

**10660: Post Processors:** The number of decimal places for feedrates in Turning posts has been modified to a more sensible default. Before if using feed per revolution, the default settings resulted in no feedrate being output without a post processor setting change.

**10662: Toolsheets:** Removed option to generate a toolsheet with VRML images.

**10670: Graphics:** When in Turning mode boundaries are now not clipped by the clipping plane for ease of use.



**10674: Toolsheets:** Fixed the Classic Toolsheet toolpath table border being slightly thicker than other toolsheet types.

**10678: Stock Model:** There were occasions when an open stock model with a drilling cycle would not be created correctly, this has now been fixed.

**10680: User Interface:** Fixed a problem the process manager toolbar not always updating correctly.

**10684: Geometry:** An exception could be thrown when trimming surfaces with boundaries. This has been fixed.

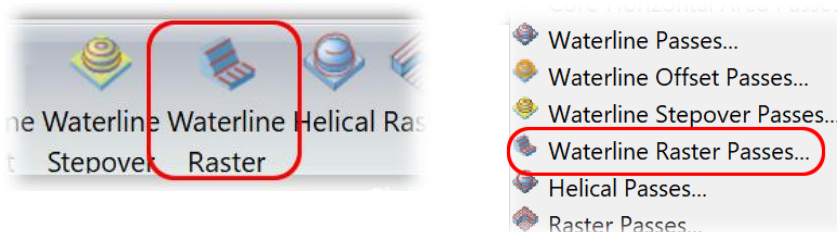
**10689: Post Processing:** It was possible for Ramp Control to be enabled for non Adaptive Clearance plans when post processing, if the remember dialogs option was enabled, this has been fixed.

## We are pleased to release NCG CAM version 20.0.03

This point release fixes a few issues that have been detected, there are also a number of worthy additions:

The **ramping moves** have been improved for **axially offset passes** and **passes along boundaries**. A new option is available making it easier to **combine waterline and raster passes**. The **helix ramping** at the **adaptive roughing** can now be performed in a **conical shape**. A new drop-down menu in the **cutter simulation dialog** makes it possible to **position a cutter directly at a collision point**. A new icon in the **ribbon** menu allows users to quickly change the **graphics background colour**. **Teeslot cutters** with **chamfers** can now be used for **deburring**. Furthermore, this version supports importing **SolidWorks 2026** files.

**536: Machining:** Added a new option for **Waterline/Raster** passes to allow steep areas to be machines as waterline passes and shallow areas as raster passes. This is done in a similar manner to Perpendicular Raster passes, as it creates two passes / toolpath folders.



### Mini Guide to Waterline Raster passes..

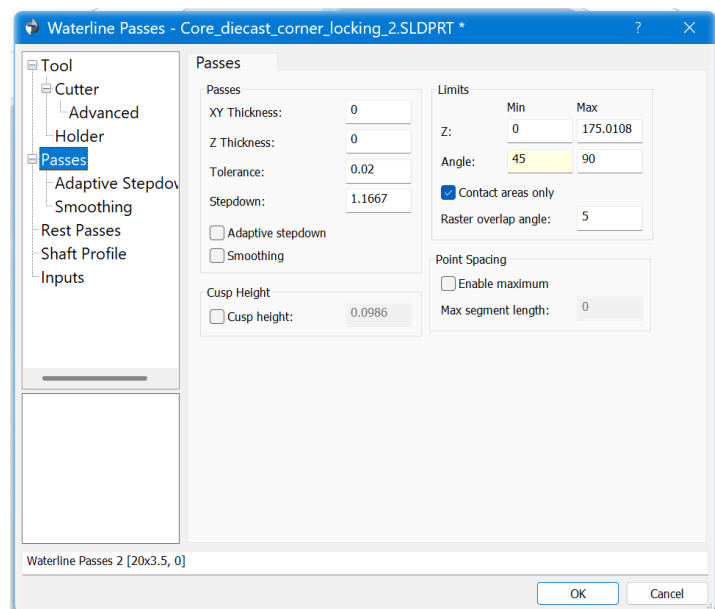
As a mini guide, this assumes you are familiar with Waterline passes and Raster passes and their parameters as individual passes.

With the required Surfaces and Boundary folders selected, **Waterline Raster Passes** can be picked from the Milling ribbon, or the mouse right context sensitive menu. That will display the Waterline passes dialog, shown to the right.

The **Min contact Angle** is set to 45° but can be changed if needed.

There is also a new option **Raster overlap angle**, and this defaults to 5°. This is how much the waterline passes and raster passes will overlap once calculated.

**Note:** In the standalone Waterline passes dialog the Raster overlap option will be seen, but is greyed out.



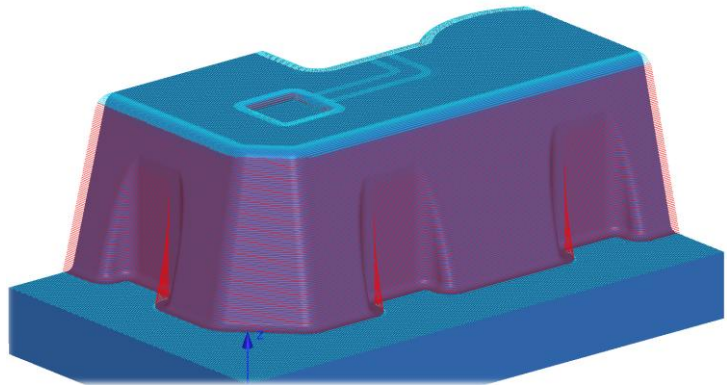
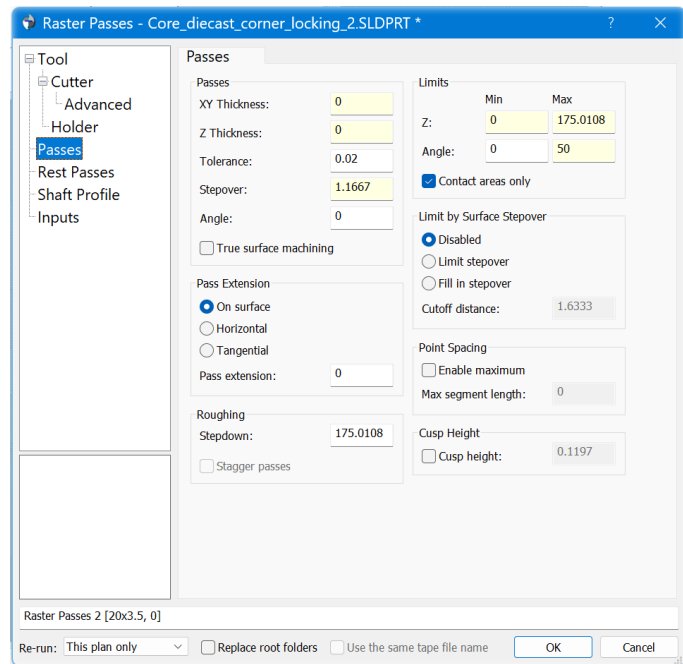
Once the OK button is pressed for the Waterline passes, the Raster passes dialog will be displayed and go to the Passes page by default, as shown to the right

It assumes you want to use the same cutter, but you can go back to the **Cutter** or **Tool** page and change it if needed. The values for **Thickness**, **Tolerance** and **Stepover** are automatically set to the values specified at the Waterline passes, and the same boundary is used.

The parameters available are the same as you've seen of raster passes before.

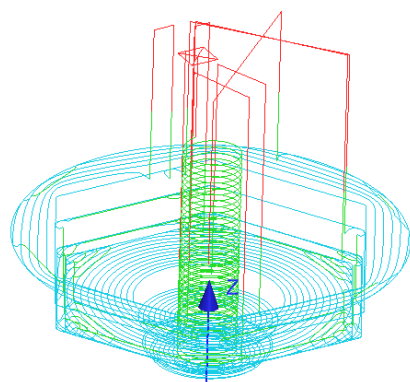
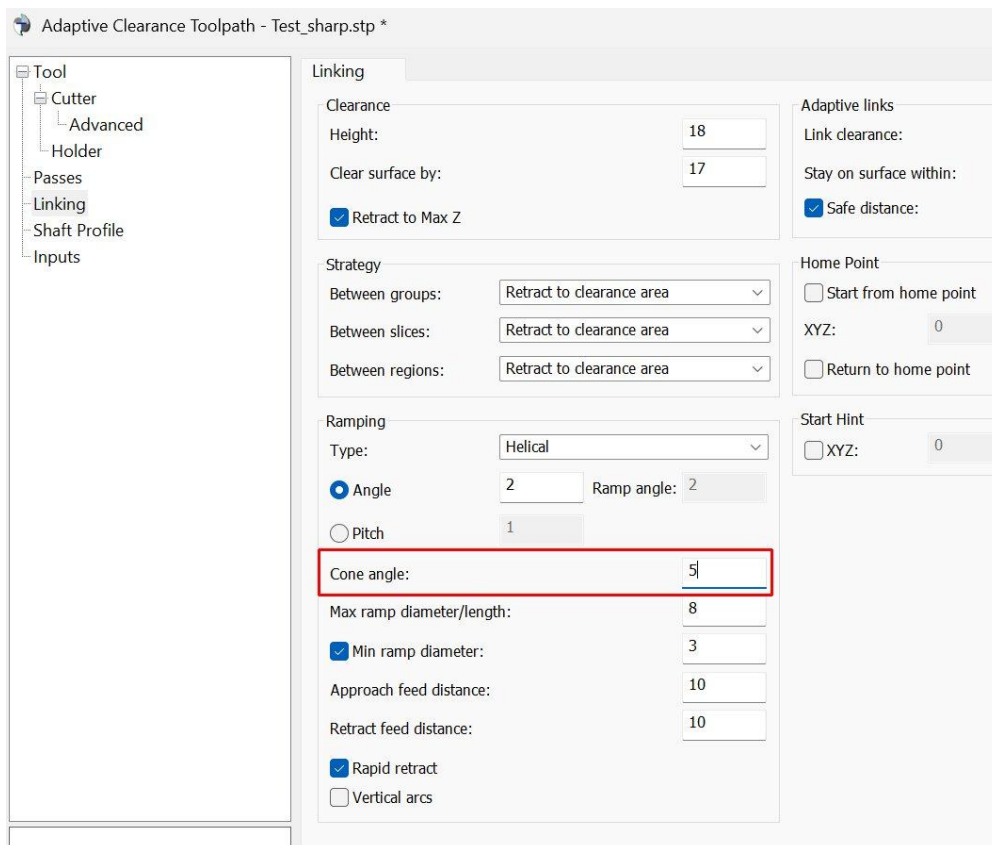
However, the **Max contact Angle** is 50°. Or more accurately it is the Waterline passes **min contact Angle** + the Raster **overlap angle** from the Waterline passes dialog.

This is the result, the waterline passes have been highlighted in red, while the raster passes are in the normal cyan colour.

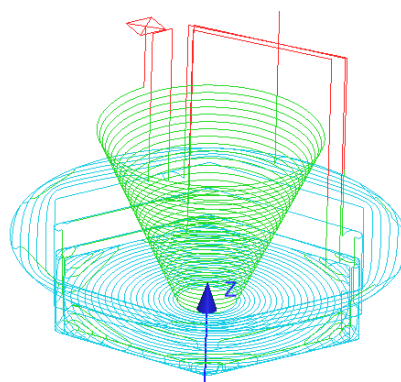


**9022: Cycles:** When a **Bore Milling Cycle** loaded a tool from the **tool database** or **MRU** list which had a **thickness** defined, that thickness was set in the bore milling cycle (Cutter > Advance), the thickness is now forced to zero.

**10207: Adaptive Clearance:** A new option for a **Cone angle** on helix lead moves for Adaptive Clearance. Is available. The **Cone angle** is available via normal **Linking** page. It would allow a conical helix in move with a few degrees of taper to provide clearance between the flutes of the cutter with the side of the entry hole just above the cutting.



Cone angle 0°



Cone angle 25° (for clarity)

**Note:** When using a **Cone angle** for plunging, **no Spindle Speed Optimisation** for the helix move is available at the **Cutting Parameters** dialog.

**10207: ModuleWorks:** The libraries have been updated to **MW2025-12**.

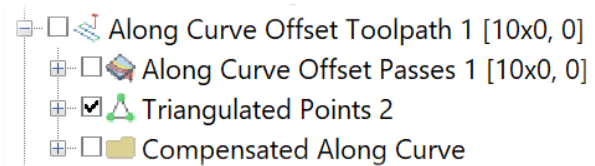
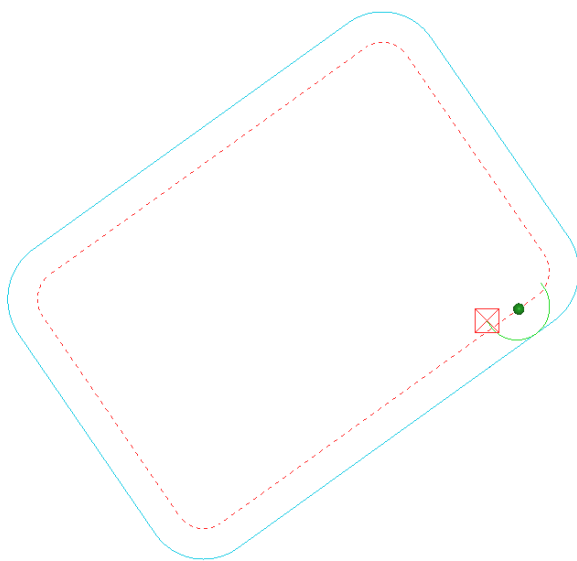
**10514: Pencil Passes:** A problem has been fixed where passes might not go to the specified depth.

**10555: User Interface:** Made the error message more informative when **failing to create a picture** (e.g. for tool sheet).

**10558: Hole detection:** Due to a rounding error in the hole depth, it was possible for the same hole to be being included in two Drilling Data folders. The rounding logic for hole depths has now been adjusted to prevent this.

**10562: Post Processor:** The wrong message was being output if Centre Drill Cycle was not supported in the macro, this has now been fixed.

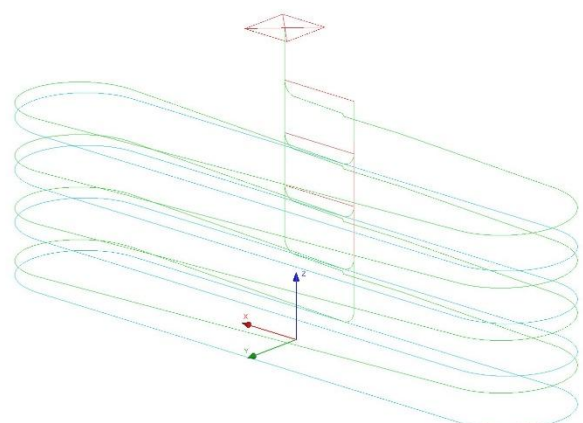
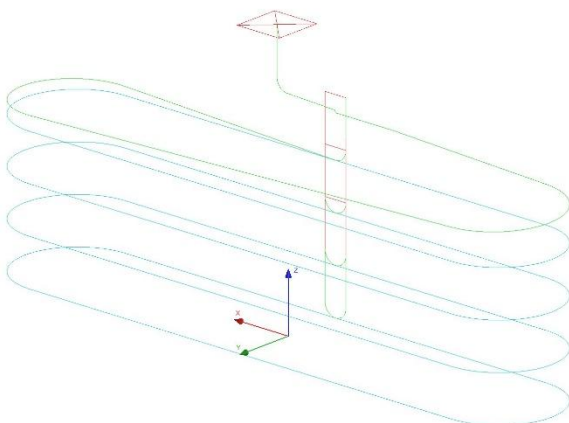
**10565: Linking: Along Curve Offset:** There was a problem which meant input **Start points** were being ignored for along curve offset toolpaths. This has been fixed.



Along curve pass, linked with a start point to control the start position of the toolpath.

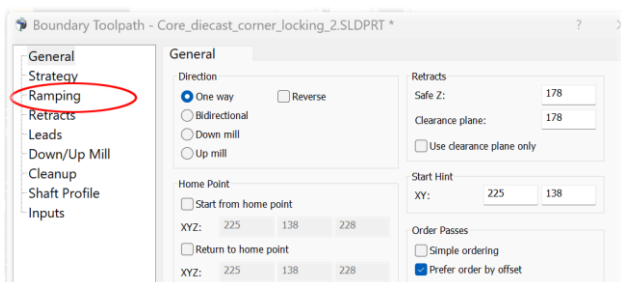
**10567: Linking: Profile ramps** were being applied **inconsistently** for **axially offset passes**. This has now been fixed.

The image on the left shows the **behaviour in versions prior to v20.0.03**. The image on the right shows the **v20.0.03** with a **ramping move** on each level.

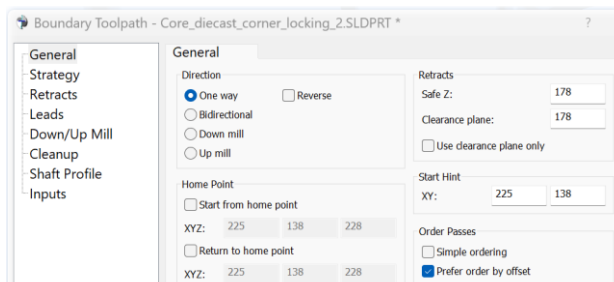


**10571: Linking: Rest Roughing Passes:** In rare situations it was possible that **toolpaths gouges** could occur when passes were linked out of order. This has been fixed.

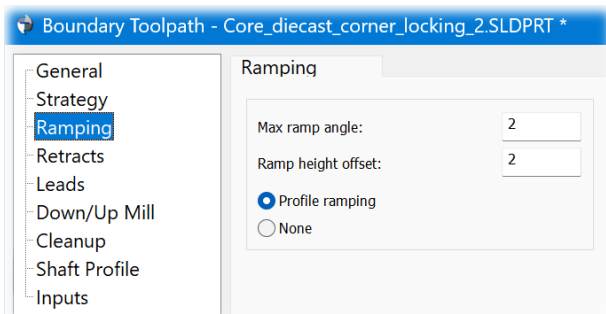
**10573: Linking: Boundary Passes:** The **Ramping** page is now accessible for **axially offset boundary passes** in the same way as for axially offset pencil passes.



NCG CAM v20.0.03

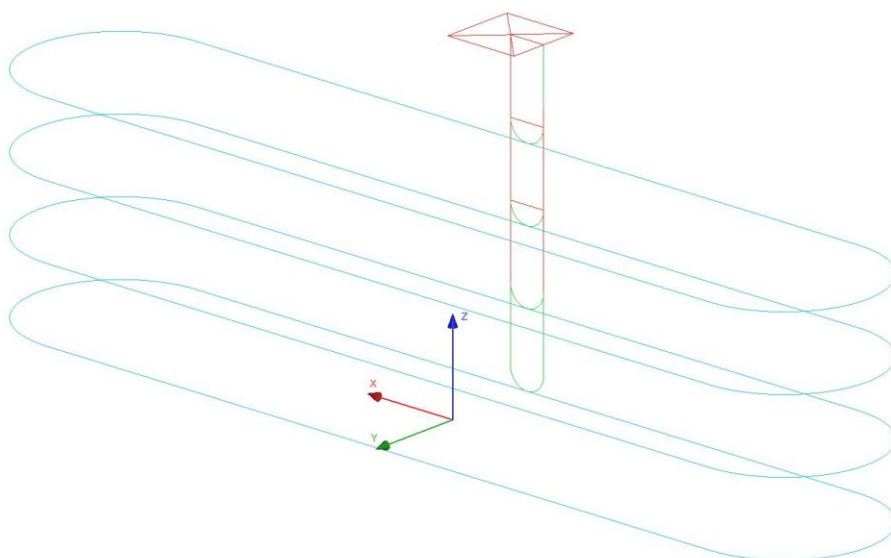


NCG CAM v20.0.02



The dialog allows the user to specify whether the plunge should be performed at an angle - **Profile ramping** - or vertically - **None**. The **Ramp angle** can also be defined.

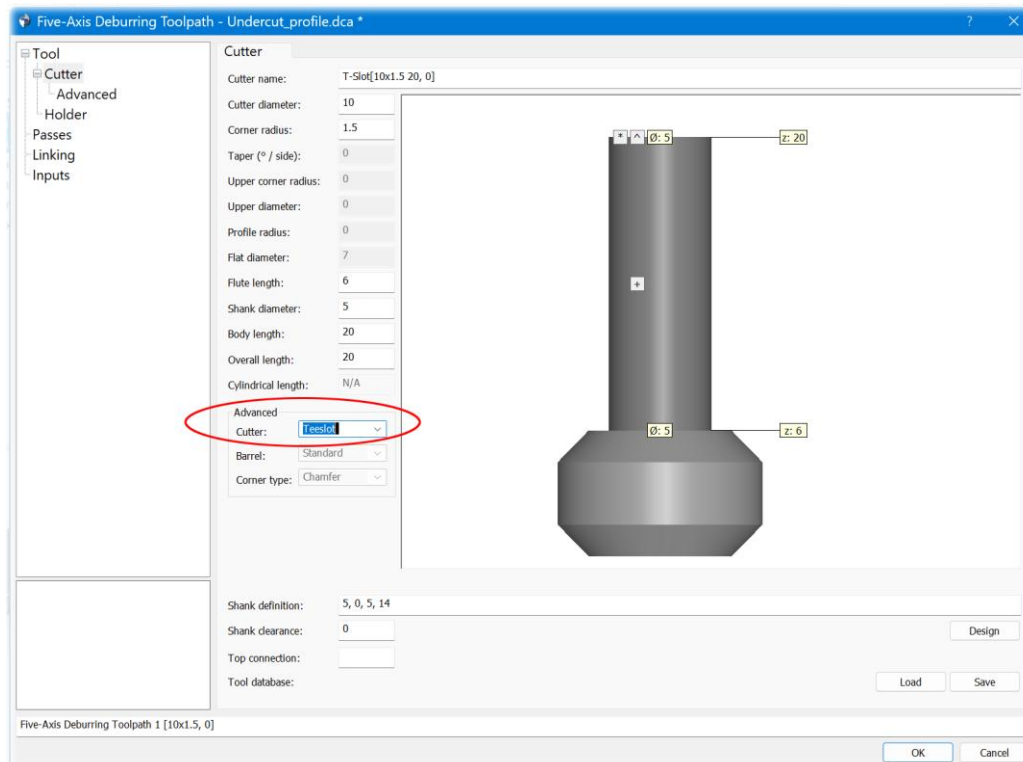
In the example shown below, the option - **None** - was used. In this case, the plunge is performed vertically for each level, with or without a vertical lead radius, depending on the setting.



**10574: Cutter simulation:** If the **column width** of the collision list was adjusted in the material removal simulation dialog, the setting was not saved and was reset when the software was closed. The changed **column width is now saved**.



**10578: Five-Axis and Three Axis Advanced:** The Deburring plan now allows the use of a chamfered Teeslot cutter.



**10581: Cutter Animation:** Changing the **toolpath display type** in the dialog when the toolpath is paused did not update the display straight away, this has now been fixed.

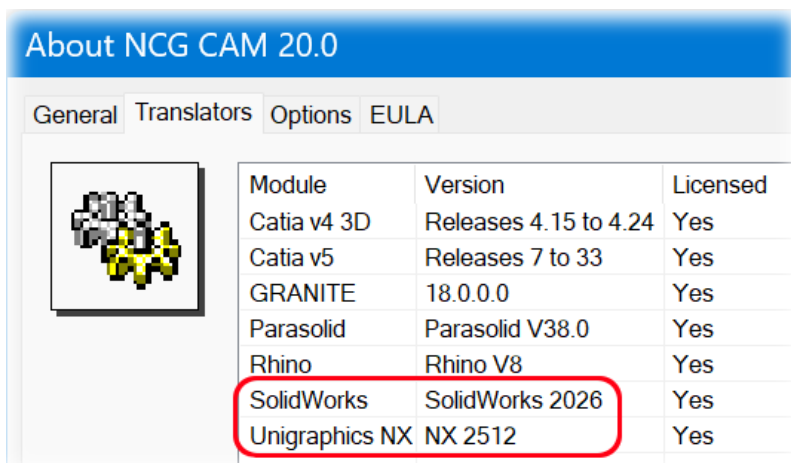


**10584: User Interface:** The **Parameter Space** option would crash if the selected surface was a raw surface (STL etc), this has now been fixed. A raw surface is no longer allowed as input.

**10585: User Interface:** The option to **save a tool to an XML format** was not handling **Teeslot cutters** correctly, this has now been fixed.

**10586: User Interface:** An error message is now shown if the plan fails when attempting to create a **2D stock model** from an **empty toolpath**.

**10593: Import: Datakit libraries** have been updated to **2026.1**. Supports **SOLIDWORKS 2026** and **UG NX 2512 Series**.

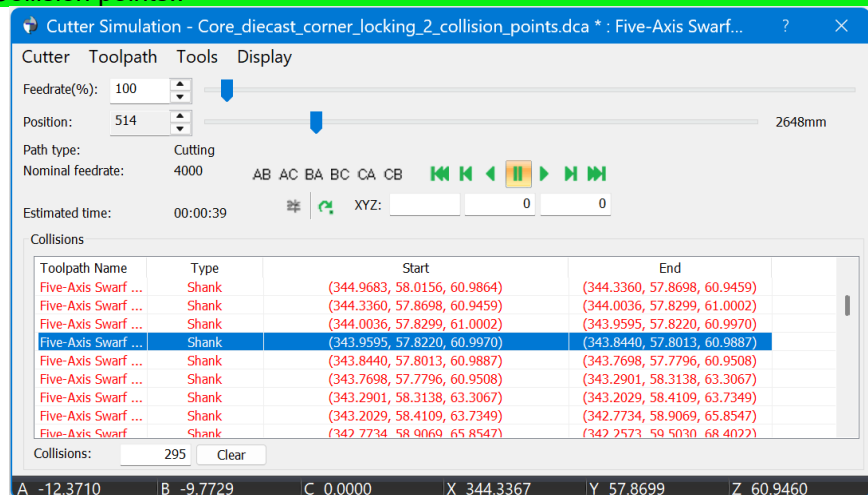


**10605: User Interface:** Fixed a problem with **autosave macros not being generated** in version 20.0.02.

**10609: Cutter Simulation:** A context menu has been added to the Cutter Simulation collision list, to add options to jump to the Start or End of a collision point. There is also an option to copy the text of the selected collision data into the text clipboard.

### Mini Guide to Cutter simulation Collision points..

In the Cutter simulation, click on the collision point you are interested in.

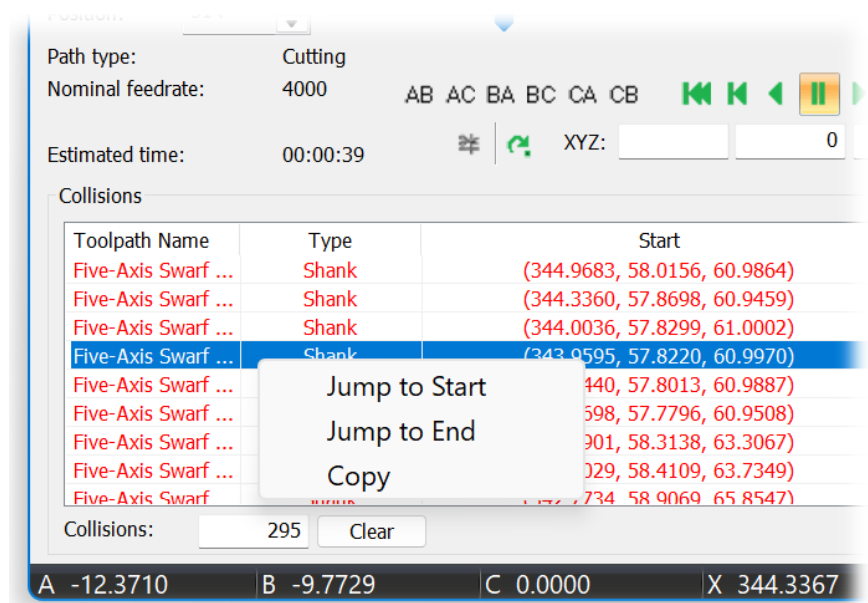


Click mouse right, and then you can pick: **Jump to Start**, **Jump to End**, or **Copy**.

**Jump to Start**, will jump to the Start coordinates of the selected collision point.

**Jump to End**, will jump to the End coordinates of the selected collision point.

**Copy**, will copy the selected line to the buffer and can then be pasted into something else.



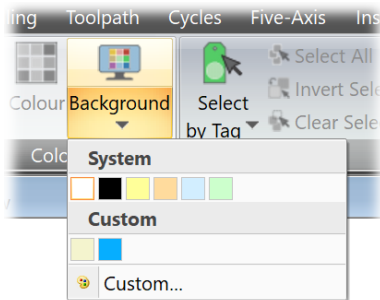
**Note:** The actual collision does not necessarily occur exactly at the displayed points, but rather between the specified start and end points.

The coordinates displayed in the collision list do not correspond to the original points of the toolpath. They are generated by the simulation along the toolpath. Otherwise, for example, a long straight line would only have one point at the beginning and one at the end.

**10610: User Interface:** The Five-Axis and Three-Axis Advanced ribbon tab were incorrectly displayed in Turning Mode, this has now been fixed.

**10612: Curves:** The Convert to Curves was not always transforming geometry correctly, this has now been fixed.

**10614: User Interface:** A button has been added to the **Edit ribbon** to set the **Background colour**. There are several 'system' colours that could be selected, or a 'custom' colour can be defined by the user.

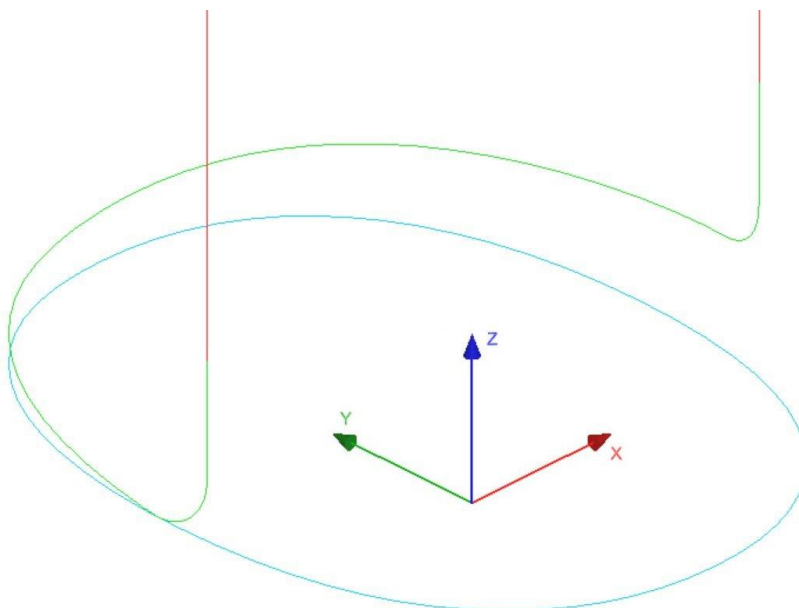
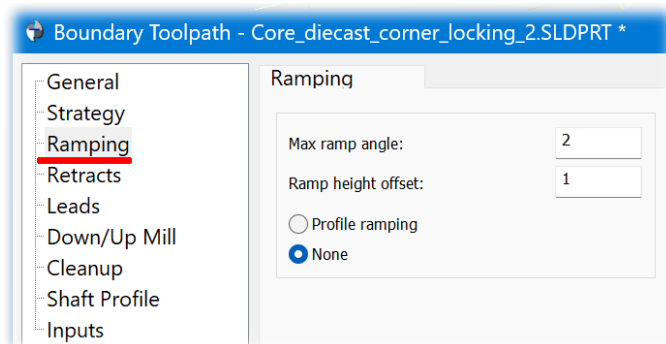


**Note:** Custom colours cannot be saved via the button in the ribbon menu. If you want custom colours to be permanently available - like the two colours in the image above - you must add them under **Options > System Colours > Background Colour**.

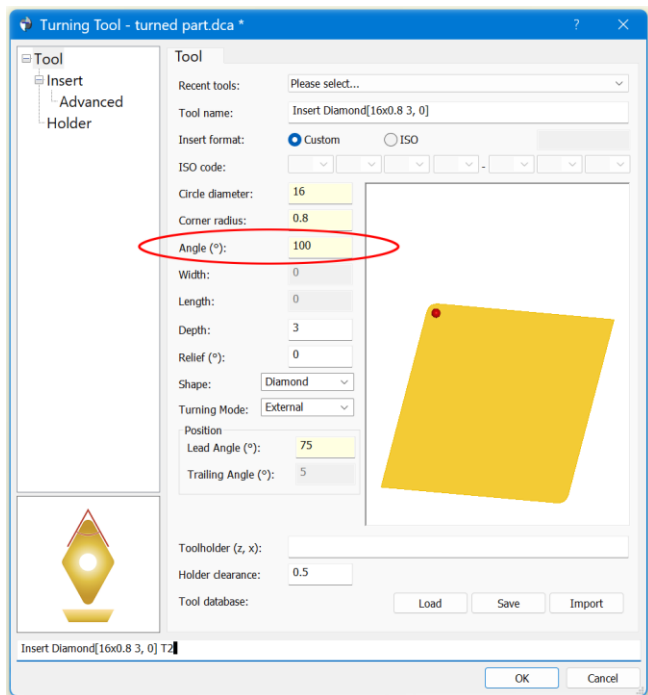
**10617: Linking: Pencil Passes:** Fixed a problem which meant surface **ramping moves** were not being created where they should be.

**10622: Linking:** The default strategy for **Along surfaces moves** has been changed from **Straight line** to **Spline Curve** for Axially Offset Pencil Passes on the **Strategy page** of the linking dialog.

**10624: Linking:** The **Ramping** page has been added to the Along Boundary linking dialog, which makes it now possible to perform a **Profile ramping** for non-axially offset boundary passes.



**10625: Turning:** Diamond insert shapes now support an insert angle greater than 90 degrees.



## We are pleased to release NCG CAM version 20.0

These release notes cover the versions 20.0.0 to 20.0.02.

Version 20 is a major release so has several new and important features. The calculation of the pencil passes has been revised and is now calculated more precisely and faster. XY offsets and stepdowns can now be defined for passes along curves. The gouge protection for drilling has been enhanced, it is now possible to differentiate between flute, shank, and holder.

A new measurement function for angle and line measurements is available. A deburring module has also been added, which is part of the Five-Axis option. Users without the Five-Axis option can purchase a Three-Axis version of the deburring module. In addition to external turning, internal turning has been added.

As a major release, a new licence code will be required.

### Database:

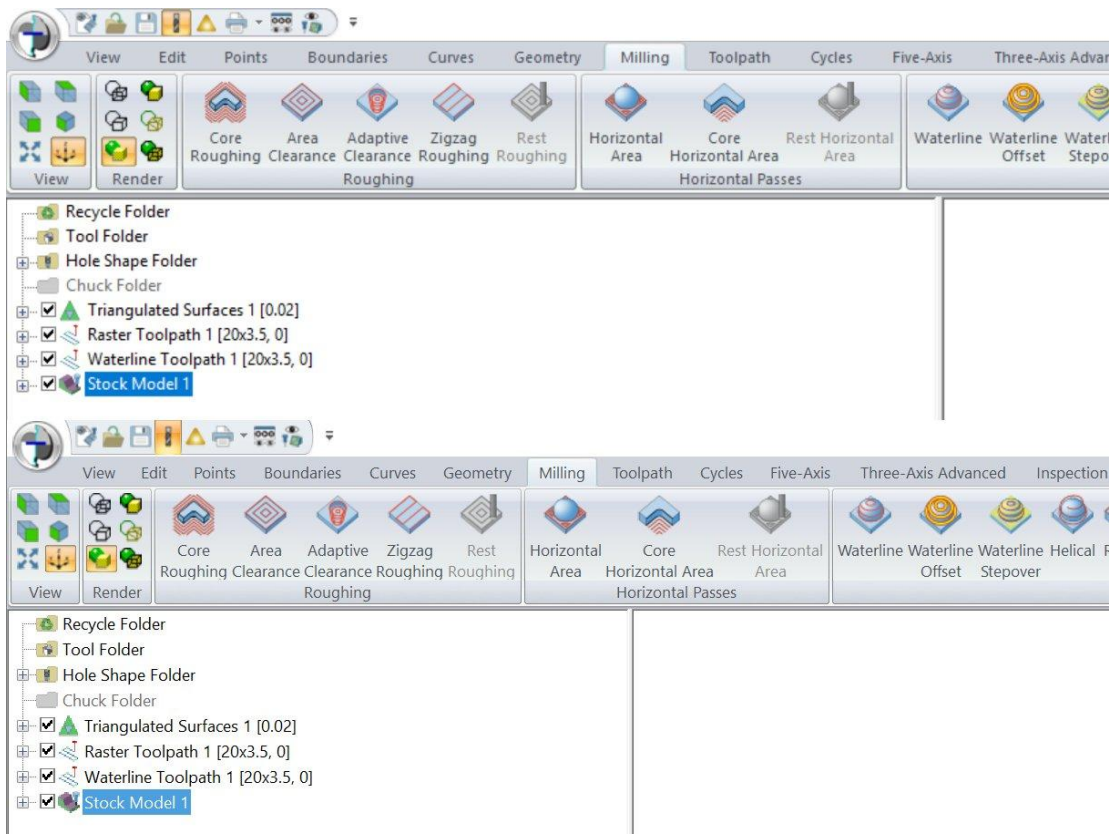
**8522: Database:** Support for loading databases (.dca files) created in versions older than v13.0.01 has been removed.

**Note:** If you need to open a .dca file earlier than v13.0.01, that .dca file should first be loaded and saved in a version between v13.0.01 and v19.0.14, then it can be opened in NCG CAM v20.0 onwards.

### User Interface:

**10404: User Interface:** Version 20 offers a new option for improved display on 4K monitors. When the display scaling is set higher than 100% - which is usually the case with 4K displays - the user interface is now rendered at the normal resolution. This makes text, in particular, appear significantly clearer.

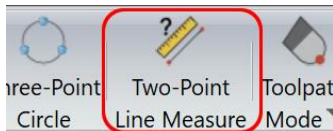
The option is disabled in the upper part of the image below and enabled in the lower part.



The option can be activated under **Options > Preferences > General**.

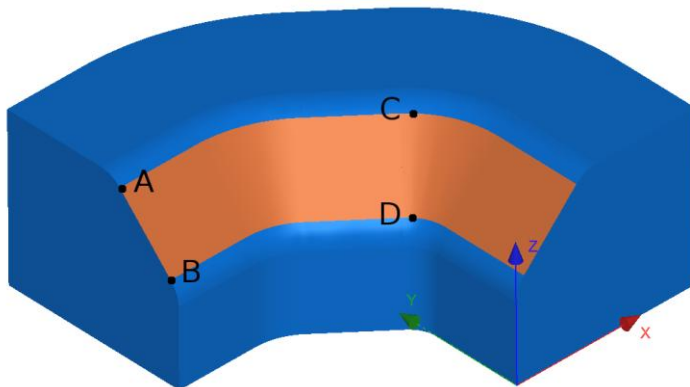


**9895: User Interface:** There is a new **Two-Point** measure function to measure distance and angle.



The previous measuring function only worked on surfaces, the new measuring will work on surfaces, curves and passes, as well as providing more information, and more precise results.

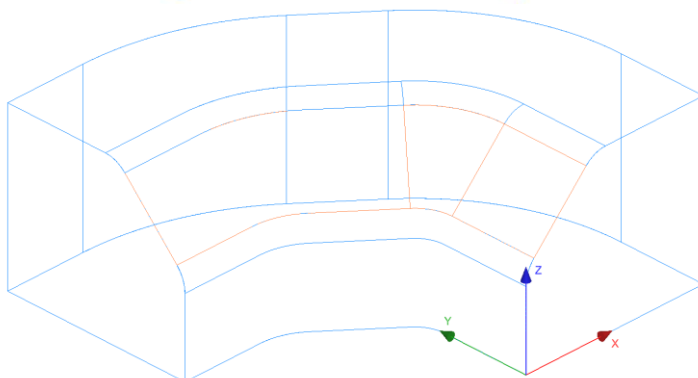
#### Mini guide to the new measure functions.



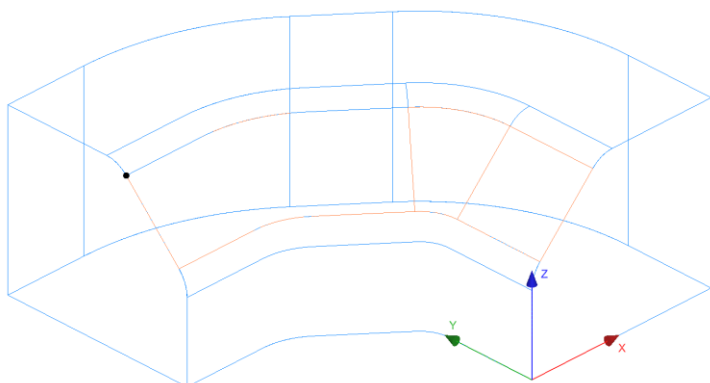
This is the example part, it is the angle of the orange surface that we will measure.

Measuring between A to B and C to D

The item to be measured (surface, curves, passes) needs to be selected in the contents tree first.



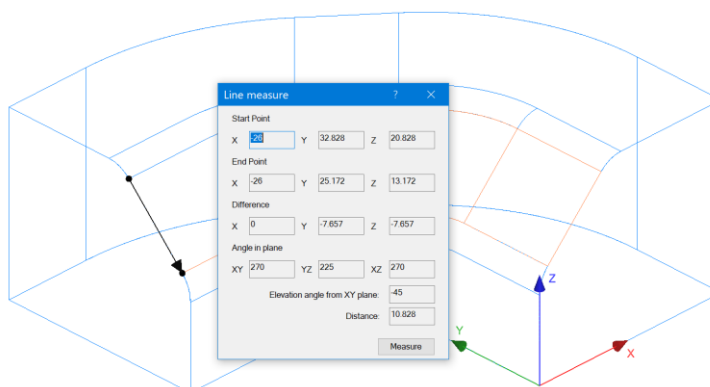
Clicking the measure button will automatically change to a wire frame view if a surfaces folder is selected.



Place the cursor on the wire framed where you want to measure from, in this example A, and click mouse left.

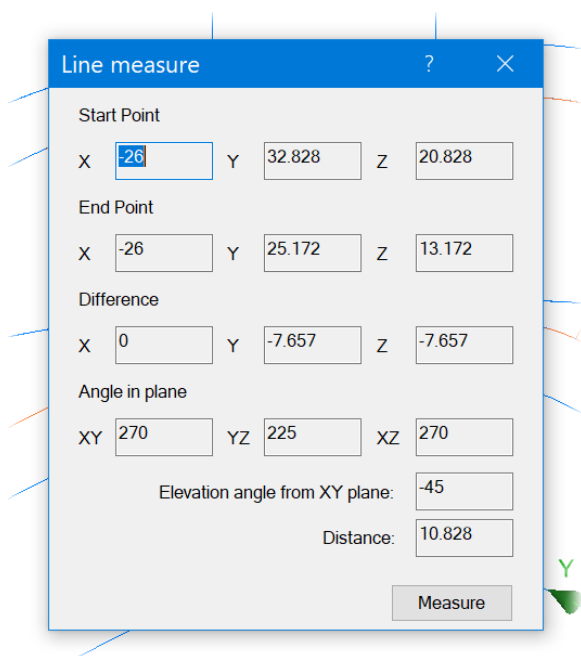
A 'dot' will be shown at that position, on the wire frame, or whatever item type is selected.

You can, if needed, now dynamically rotate the part, zoom in/out or pan as needed to get a better view of where you want to measure to



Now place the cursor at the position you wish to measure to, and make another mouse left click.

An arrow will be shown between where you are measuring from and to, and a dialog will be displayed with details of what is being measured.



Looking at the information in the dialog:

There are the XYZ coordinates for the **Start Point** (A), and then the XYZ coordinates for the **End Point** (B).

The **Difference** is the calculated, subtracting the **Start Point** from the **End Point**.

The **Angle in plane** gives the angles in the **XY**, **YZ** and **XZ** planes.

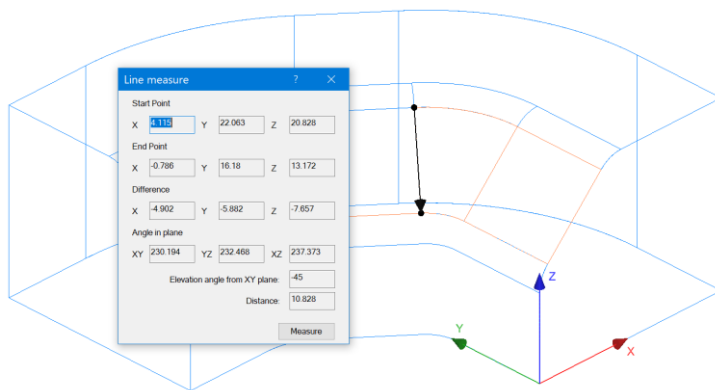
**Elevation angle from XY plane** gives the angle relative to the XY plane

The **Distance** is the straight line distance between the two points.

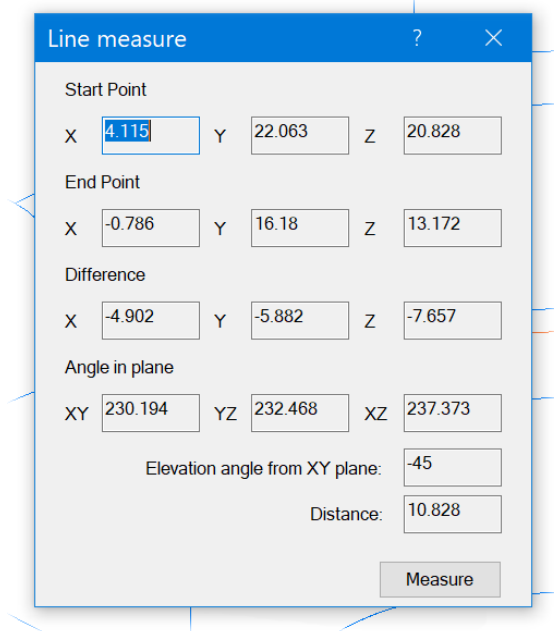
It is possible to select the value on any of these boxes and copy it to the Windows copy / paste buffer.

Clicking **Measure** assumes you want to do another measure between two more points and keeps in the Measure mode.

If you have done measuring, click the **X** in the top right of the dialog and exit from the measure mode.



Repeating the first few steps above, but measuring between points C and D

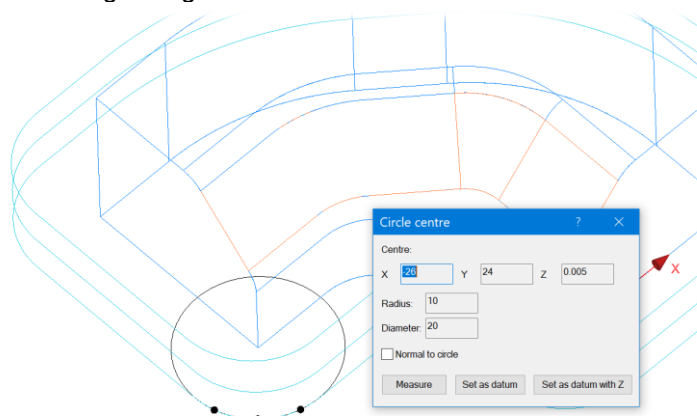


Because the planes being measured on are not the same as the primary axis (XY or Z), the **Angles in plane XY, YZ and XZ** values may not be so helpful, despite being correct.

This is when the **Elevation angle from XY plane** can be helpful, which shows an angle of -45 degrees.

### 3 point circle

There are some little changes to the existing **3 point circle** measuring function, to make the two measuring dialogs more consistent with each other.



In the image left, a 3 point circle has been created on the waterline passes.

It will display a slightly different dialog to what was displayed before NCG CAM v20.0

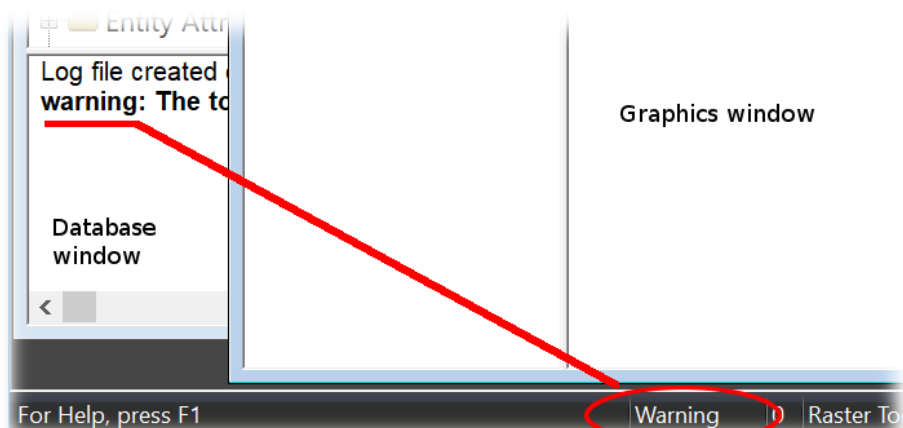
The same information and options are there as before, other than **OK** has been replaced by **Measure**.

As with measuring between points, clicking **Measure** assumes you wish to measure another radius, if you want to Cancel, use the **X** in the top right of the dialog. It will close the dialog and deactivate the measurement functionality. In addition the graphics is reset to the mode it was before.

**10394: User Interface:** Error and Warning: messages are displayed in the text field of the database window. Since the database window is usually behind the graphics window, it was often not apparent when a message was present.

With version 20, an additional notification is displayed in the status bar for error or warning messages – either as **Error** or **Warning**.

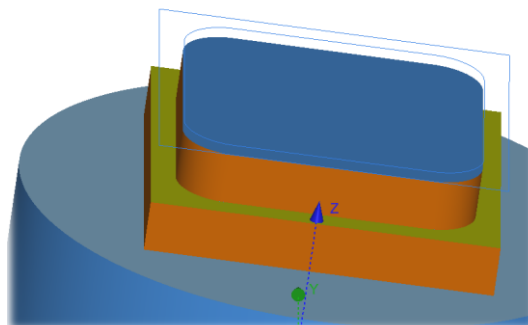
If you switch to the database window to view the message text and click in the text field, the message in the graphics window's status bar is cleared.



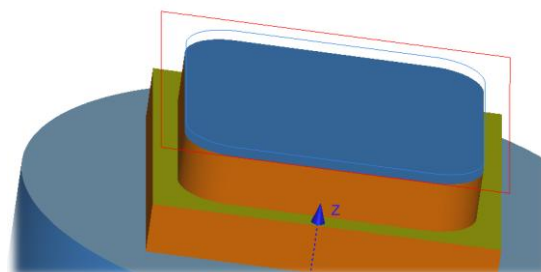
**10376: User Interface:** The **Set remembered colour** function now works for entities with more than one colour attribute (assuming they are the same colour). Using the 'Set Colour as Remembered Colour' now sets all the colour attributes of an entity (formerly it only modified the first). So it is now possible to select a curve and change its colour, select curves of the same colour.

The curves were extracted from the yellow surface, and planarized just above the part for clarity.

The NCG CAM default colour is blue.



One of the curves is selected here:



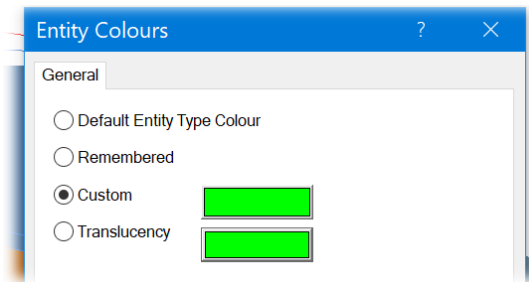
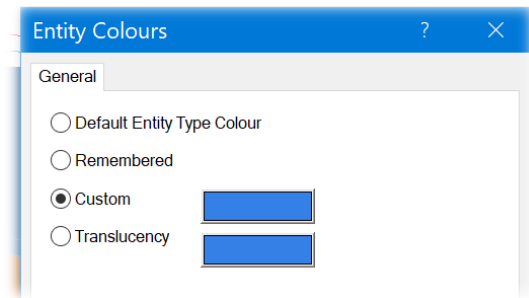
**Colour** is picked from the View ribbon or mouse right menu, and then the **Custom** option selected.

There are two colour panels, the first is for linear sections of the curve, and the second is for arc sections of the curve.

Pick the first panel, and the Windows colour palette will appear, select your preferred colour, and click OK.

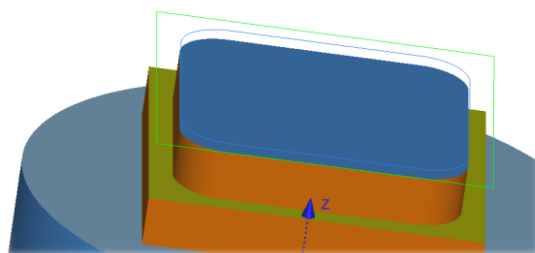
Then repeat that for the second panel. But they must be the same colour exactly – the same RGB values.

That curve is now shown the chosen colour.



Select the curve that had its colour changed.

Mouse right and pick **Remember Selected Colour**.

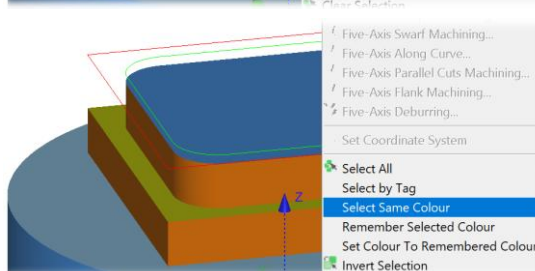
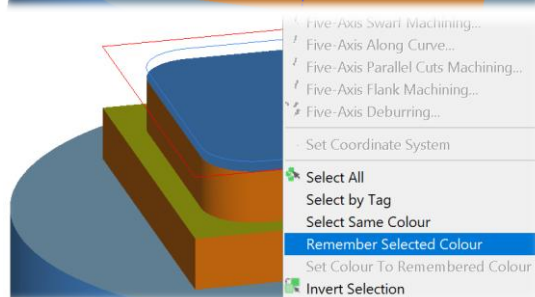


Select the curve that is still shown using the default blue colour. Or the curves to want to change to the remembered colour.

From the mouse right menu, pick **Set Colour To Remembered Colour**.

Click somewhere on the background to clear the select colour and see the curves' new colour. Select one of the curves, and from the mouse right menu, pick: Select Same Colour.

All curves should the same colour ss the originally selected curve should now be shown in the Select colour.

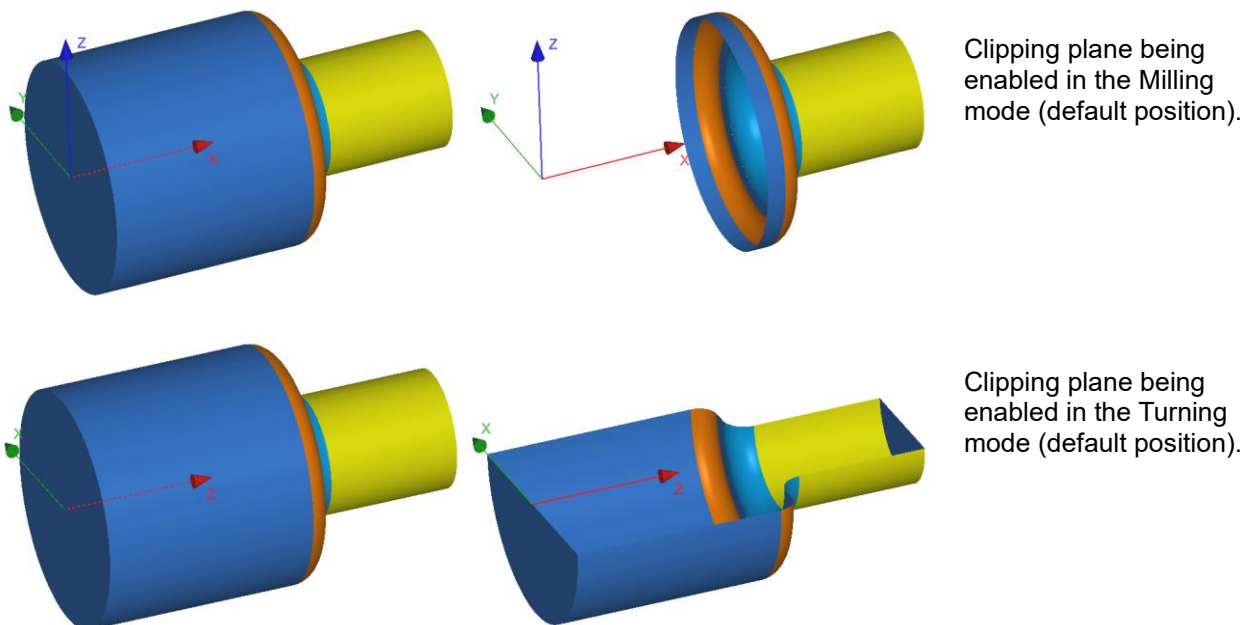


**10463, 10470: User Interface:** Estimated times for **Cutting, Ramp down, Ramp up, Rapid** and the **Total** time have been added to the length information at the **Toolpath Analyse**.

In addition it is now possible to select a **Cutting Parameter plan** as input to the Toolpath Analyse to specify the correct feedrates for the time estimation.

| General         |                                                                                                                                                                                                                                                                                                                                           |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
|-----------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------|------|----------|---------------------|------------|--------------------|----------|--------------------|--------|-------------------|--------|---------------------|
| Length and Time |                                                                                                                                                                                                                                                                                                                                           |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
|                 | <table> <tr> <th>Length</th><th>Time</th></tr> <tr> <td>Cutting:</td><td>19756.2054 00:04:56</td></tr> <tr> <td>Ramp down:</td><td>2404.3408 00:00:36</td></tr> <tr> <td>Ramp up:</td><td>2438.5641 00:00:36</td></tr> <tr> <td>Rapid:</td><td>557.0304 00:00:08</td></tr> <tr> <td>Total:</td><td>25156.1407 00:06:17</td></tr> </table> | Length | Time | Cutting: | 19756.2054 00:04:56 | Ramp down: | 2404.3408 00:00:36 | Ramp up: | 2438.5641 00:00:36 | Rapid: | 557.0304 00:00:08 | Total: | 25156.1407 00:06:17 |
| Length          | Time                                                                                                                                                                                                                                                                                                                                      |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
| Cutting:        | 19756.2054 00:04:56                                                                                                                                                                                                                                                                                                                       |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
| Ramp down:      | 2404.3408 00:00:36                                                                                                                                                                                                                                                                                                                        |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
| Ramp up:        | 2438.5641 00:00:36                                                                                                                                                                                                                                                                                                                        |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
| Rapid:          | 557.0304 00:00:08                                                                                                                                                                                                                                                                                                                         |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
| Total:          | 25156.1407 00:06:17                                                                                                                                                                                                                                                                                                                       |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
| Segments:       | 1831                                                                                                                                                                                                                                                                                                                                      |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
| Fragments:      | 429                                                                                                                                                                                                                                                                                                                                       |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
| Arc length:     | 33.8 %                                                                                                                                                                                                                                                                                                                                    |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |
| Line length:    | 66.2 %                                                                                                                                                                                                                                                                                                                                    |        |      |          |                     |            |                    |          |                    |        |                   |        |                     |

**10192: User Interface:** There are now two Clipping Planes, one for Milling and one for Turning, each being used in the appropriate machining mode. In the milling mode the default clipping plane is a YZ plane (vertical), as it was before. In the turning mode the default clipping plane is XZ (horizontal), which should be more useful when turning.



**10437: User Interface:** There was a problem when ticking and un-ticking a Machine Along Curve plan not drawing the compensation curves, this has now been fixed.

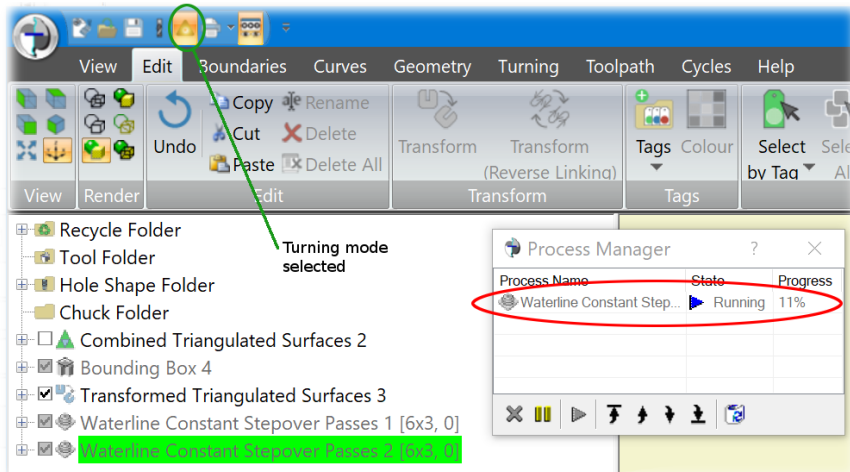
**10441: User Interface:** The Stock Model – **Z-Trim** dialog did not display the **dialog guides** correctly when a rotated boundary selected, this has now been fixed.

**10371: Graphics:** The View along tool axis (Ctrl + Shift + X/Y/Z) was not working for edited curves folders. This has been fixed.

**10413: User Interface:** Fixed a problem that could result in the 2D tool guide remaining in the graphics view after the dialog has been closed.

**10266: Curvatures:** If you selected a UV Direction Surfaces folder and attempted to do a Curvatures Analysis it would fail with no message, this has now been fixed.

**10051: Process Manager:** The Process Manager now recognises the difference between Milling and Turning plans, greying plans out when appropriate based on the selected machining mode.



**10502: User Interface:** In the context menu of a triangulated surfaces folder, the entry **3D Tool Guide** has been removed, as the menu would otherwise be too long for a screen resolution of 1280x1090.

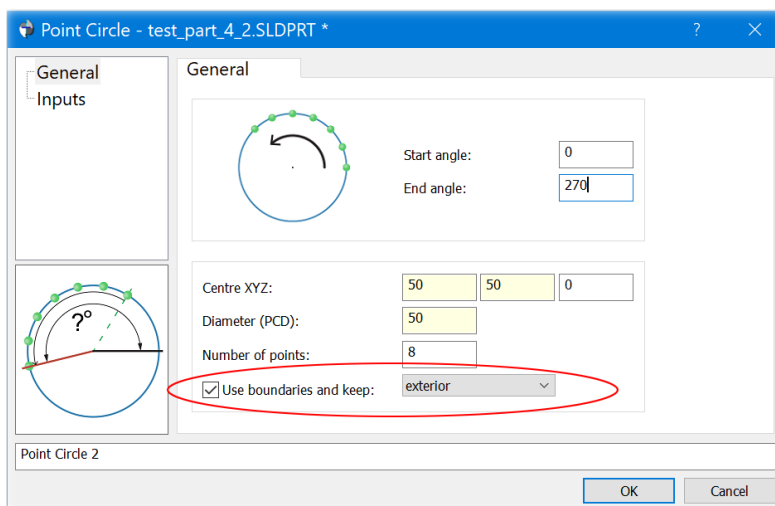
**10523: User Interface:** There was a problem with the Post Processor Dialog not updating the toolpath page correctly when the inputs have been changed, this has now been fixed.

**10525: User Interface:** There was a problem when changing the inputs of a ToolSheet plan on the dialog inputs page that could cause NCG CAM to crash, this has now been fixed.

**10526: User Interface:** Fixed databases not being saved when exiting the program.

## Geometry:

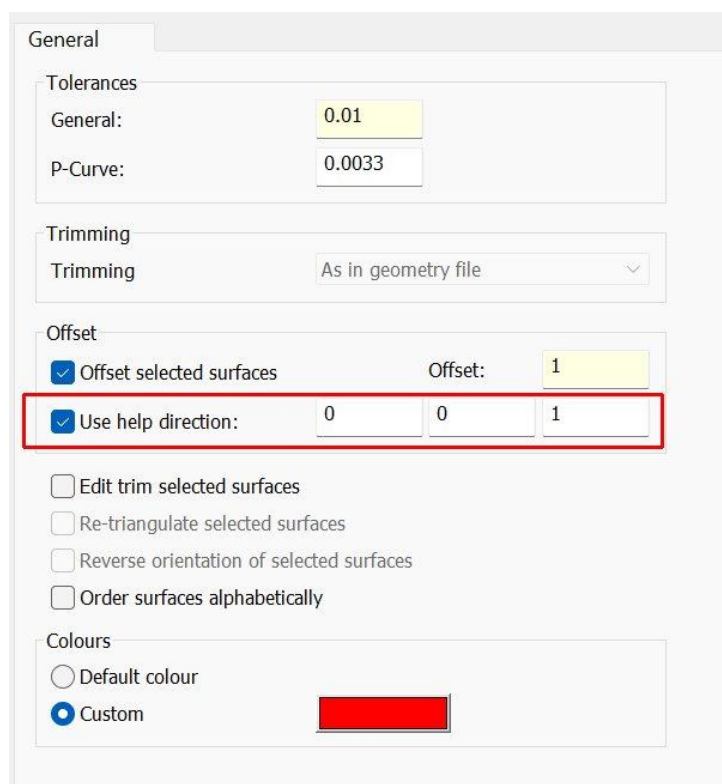
**10241: Points:** The Point Grid and Point Circle plans now allow for trimming to a boundary provided as an input to the points being created, with options to keep the Interior or Exterior points.



**10446: Extract Sliced Curves:** The validation for the resolution has been improved for the case where Under Cuts are detected. This stops a crash that could occur due to running out of memory.

**10342: Boundaries:** When creating a boundary manually, and with the clipping plane active the boundary and grid were not drawn. While the boundary is in edit mode, it will be visible even if it is on the hidden side of the active clipping plane.

**10199: Offset surfaces:** For offset surfaces, the side on which the offset is created depends on the surface direction. However, a **help direction** can also be specified as a vector. In this case, the offset is in the direction of the vector, independent of the surface direction. Applying the help direction was not working correctly and is now correct.

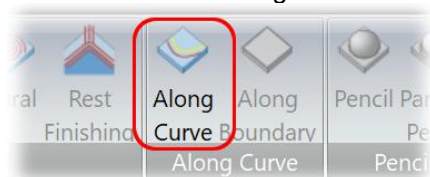


**10247: Boundaries:** It is now possible to create a boundary from a 2D Stock Model.

### Milling Passes:

**8298 & 10494: Along Curve Offset:** A new strategy Machine **Along Curve Offset** has been added. In the User Interface the **Along Curve Passes** have been replaced with **Along Curve Offset Passes** in the Milling ribbon and right mouse context menu.

The button and wording in v19.0.15

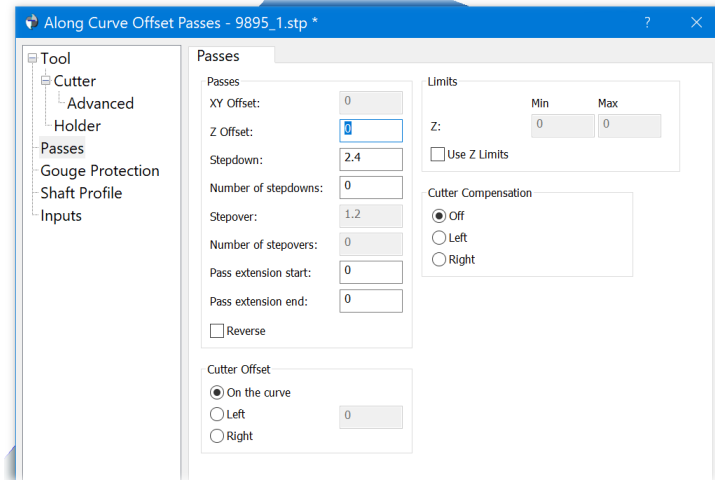


There is a minor change to the button and wording in v20.0.01



The same button and dialog allows for the original Along Curve pass and the new Along Curve Offset passes. The default settings **Number of stepdowns** and **Number of stepovers** will give a **single** pass just as the along Curve passes did previously.

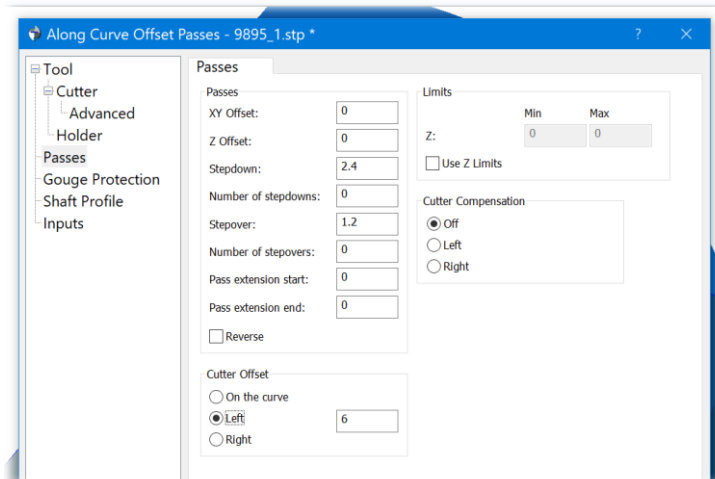
## Mini guide to Along Curve Offset Passes.



Like the pre-existing along curve passes, the required surfaces and curves folders need to be selected in the contents tree. The curve(s) however must be planar, if offsets are to be generated.

Initially, then dialog opens with the **On the Curve** option selected. The offset passes is only available when the **Cutter Offset**, is set as **Left** or **Right**. That will then enable some of the other options in the dialog.

Like the Waterline Offset Passes, these passes can be useful for thin walls, allowing more supporting material to be left as it is machined.



With **Cutter Offset, Left (or Right)** set, some of the other parameters are available.

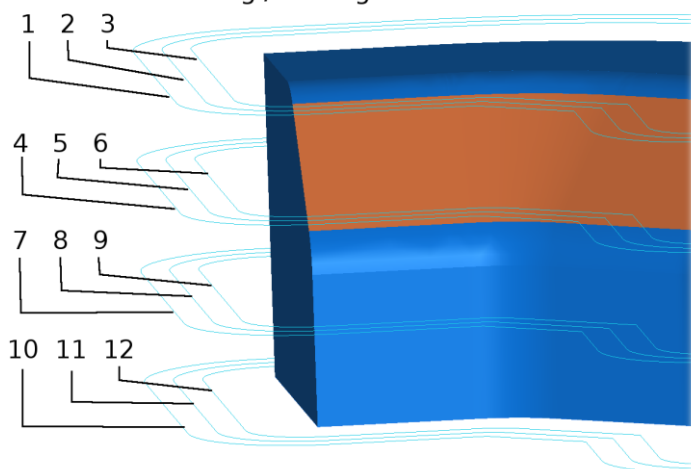
**Stepover:** Is the distance or pitch between the offset passes.

**Number of stepovers:** Controls how many offset passes are created.

**Stepdown:** This is the stepdown along the tool axis.

**Number of stepdowns:** Defines how many stepdowns to create. Note this option is available because the **Use Z levels** is not selected. If **Use Z levels** is selected, the Z range can be set, and the **Number of stepdowns** is greyed out.

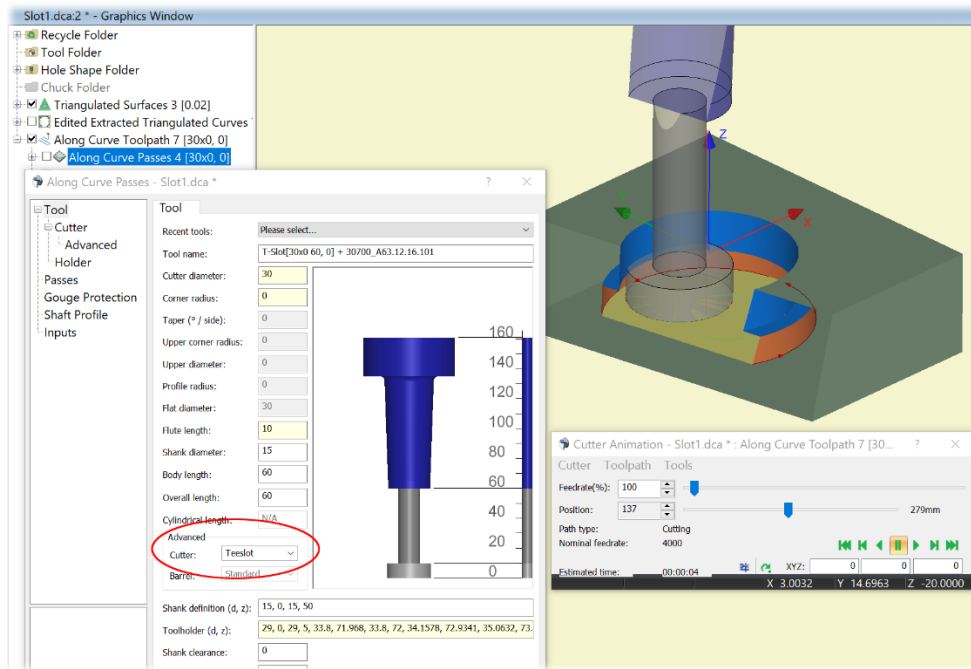
Cutting / linking order



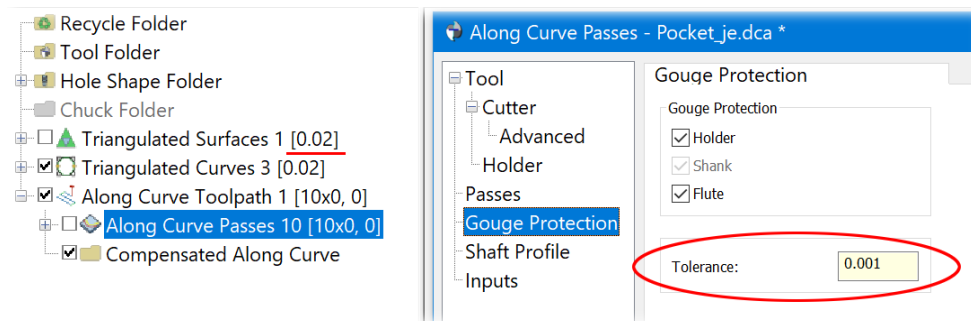
The result is similar to having created the passes at the final depth, and then axially offset them.

This shows the linking / machining order based on the setting in the previous dialog.

**10136: Along Curve Passes:** It is now possible to use **Lollipop**, **Dovetail** and **Teeslot** cutters for this strategy. Checks should be made to ensure the entry and exit moves are clear.



**10283: Along Curve Passes:** Fixed a problem with the gouge protection which prevented passes from being machined to within the applied tool clearances of surfaces.



Previously the 'Gouge tolerance' was twice the triangulation tolerance and could not be set to a lesser value. Now, while the default value is still twice the triangulation tolerance, it can be set to a smaller value if needed. If you need or want to have a greater amount of clearance for the shank of the cutter that may be possible by setting the Shank clearance.

**Note:** The Shank clearance only applies to parts of the shank that are of a greater diameter than the flutes diameter. So 8.0 dia flutes and shank, with a shank clearance of 0.25 will have no effect, but 8.0 dia flutes and 10.0 dia shank and a shank clearance of 0.25 will keep the 10.0 dia 0.25 clear of the part.

**10302: Machine Along Curve Passes:** The accuracy of the trimming used to gouge protect passes has been tightened.

**10252: Machine Along Curve Passes:** We now display the compensated passes (the red dotted line when cutter compensation is enabled) with the XY offset applied. Previously it was possible for a small cutter, and a large XY offset, the compensated pass could appear to be the wrong side of the toolpath, despite the toolpath being correct.

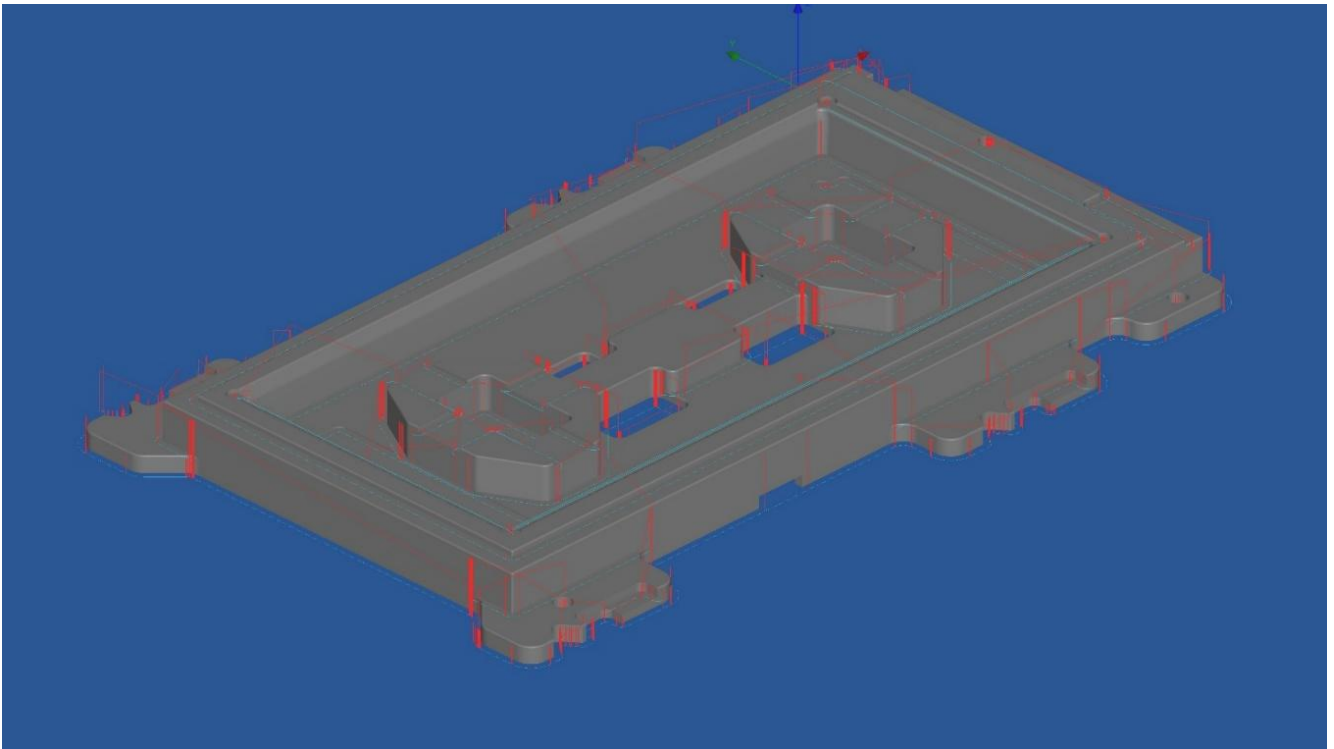
**7101: Z Profile Passes:** Improvements have been made to the Z profile style passes (Area Clearance, Core Roughing, Horizontal Area, Waterline) to reduce the number of cases that failed to machine to depth. This could be more noticeable with very small stepdowns and usually small cutters, in the database window a message like **error: Failed to calculate machining profiles at level...** would have been produced, and the plan would fail.

**9434: Pencil Passes:** The quality and calculation speed of the pencil passes have been significantly improved.

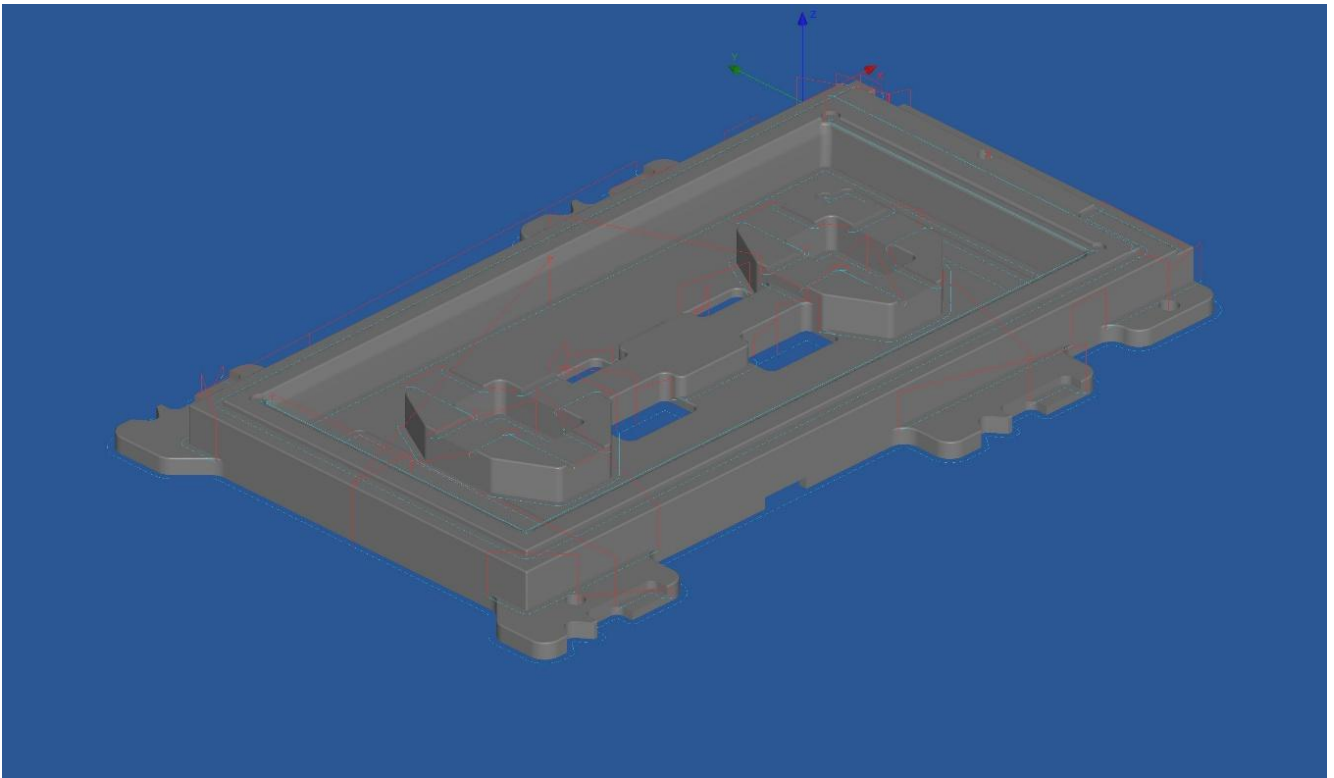
The optimization primarily addresses small gaps in the paths, which previously resulted in disjoint toolpaths.

The following images show the machining of a component with vertical surfaces using an end mill.

In v19, numerous gaps appear in the paths, which lead to many retracts in the toolpath. The result is not useable.

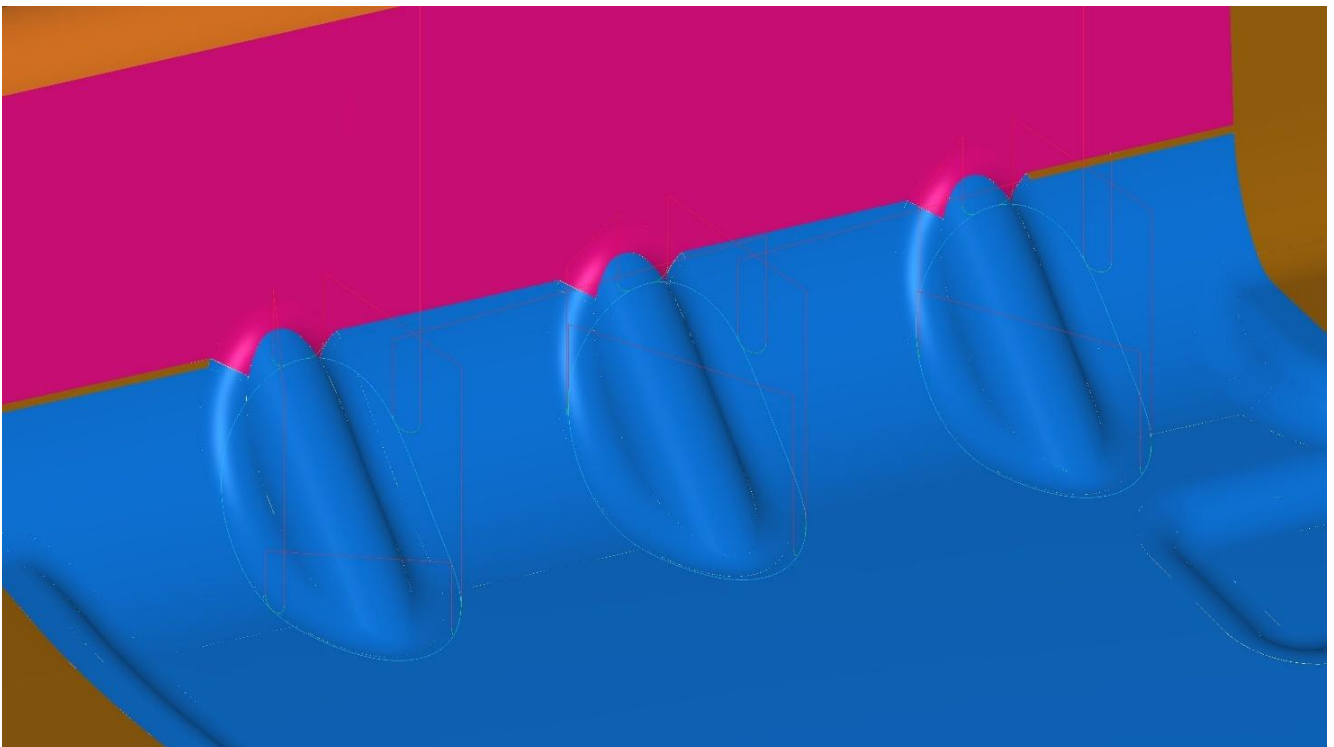


In v20, however, the toolpaths is continuous and without gaps.

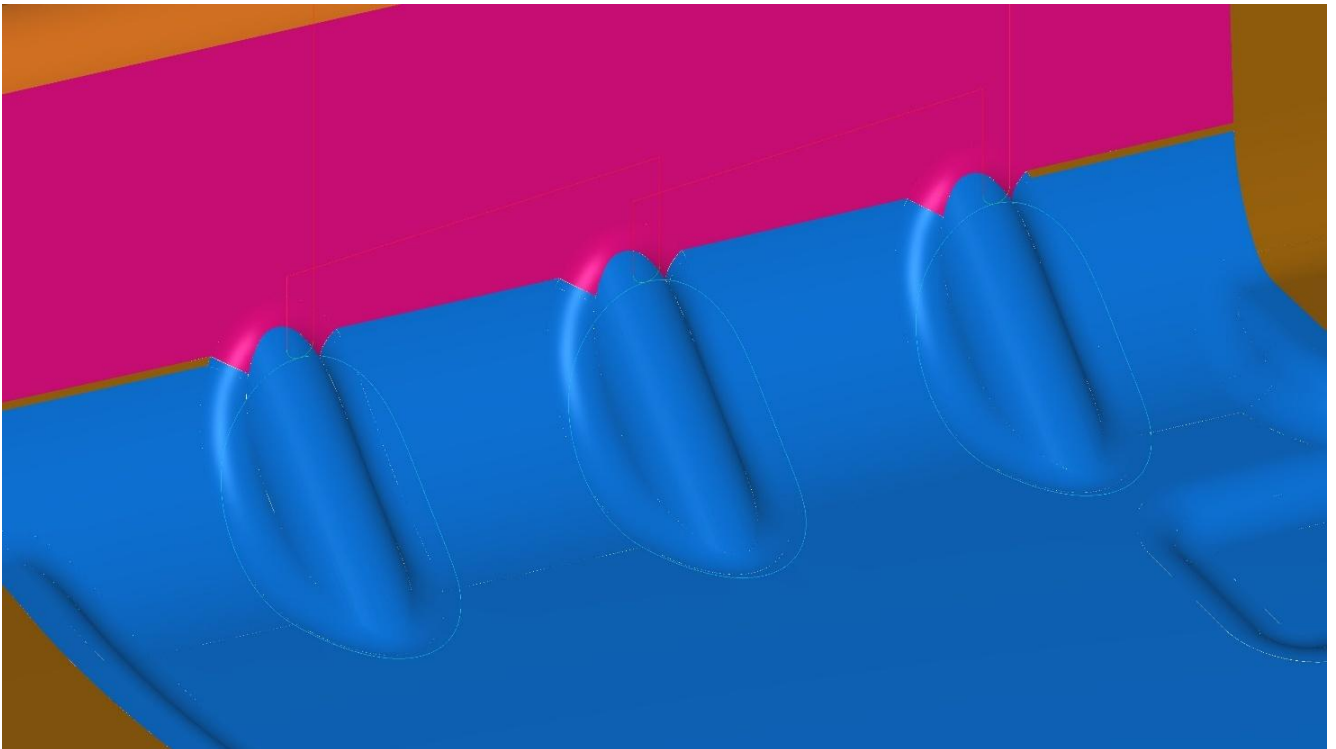


The calculation time for the pencil passes in the example above is 2min 58s in v19, and only 21s in v20.

The following images show pencil passes with a ball nose cutter. In version 19 gaps occur in the passes, causing the toolpath to be divided into multiple segments.

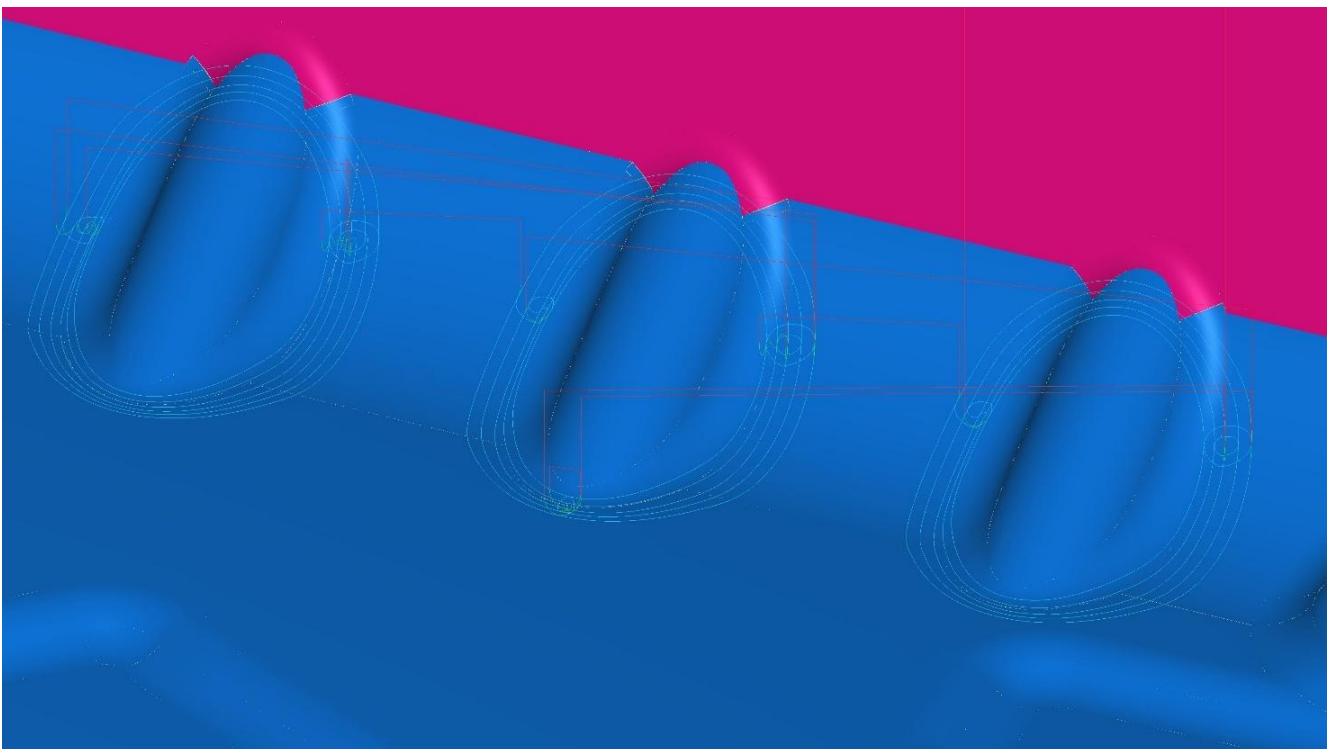


In v20 the passes are closed, and there is only one approach point per area.

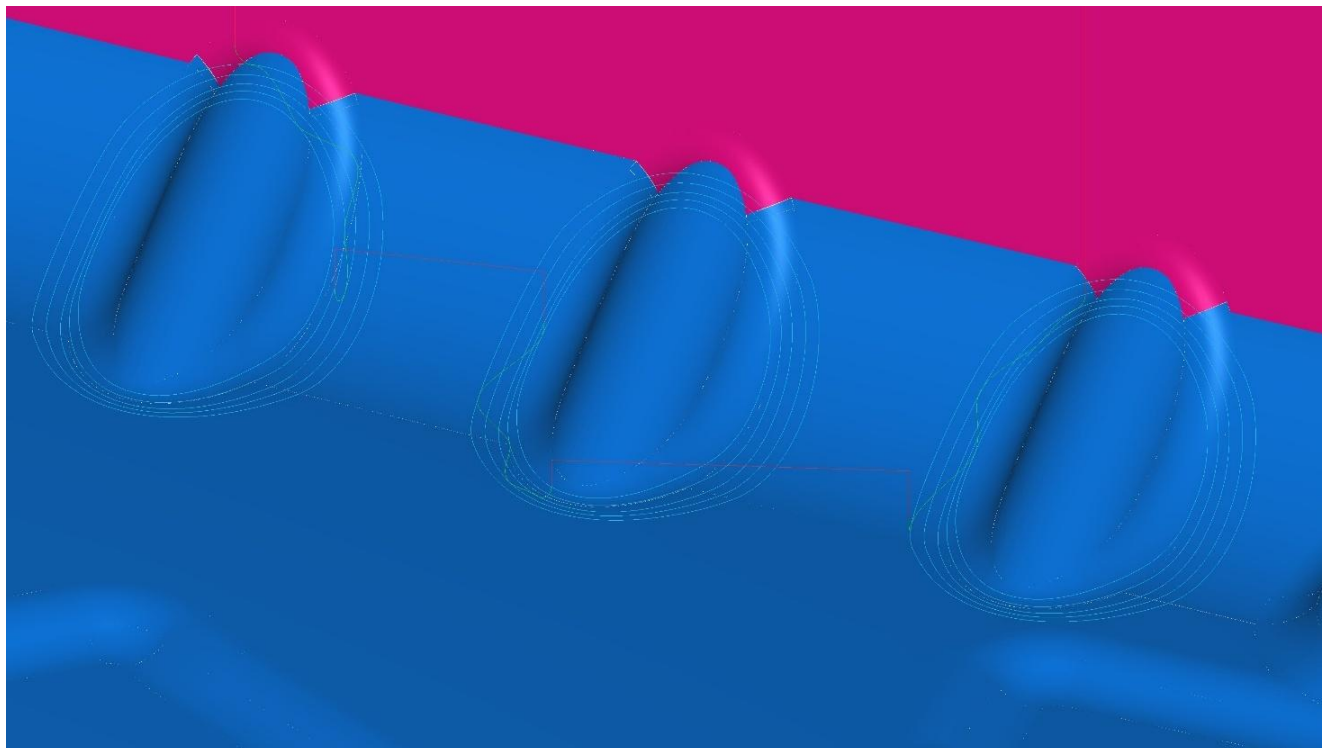


Since pencil passes are also used in the background to calculate rest finish passes, the improved pencil passes also improve the result of this strategy.

The following image shows the rest finish passes in v19. The gaps in the underlying pencil passes create spirals in the rest finish passes.



Using the same parameters in v20, the paths are cleanly connected and the result is significantly improved.

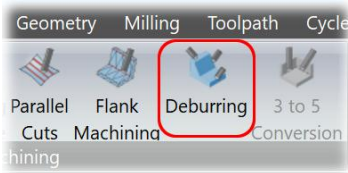


**10286: Pencil Passes:** The **Bi-tangency angle** parameter for pencil passes now has a different default value according to cutter type: **10°** for ball cutters (no change); **20°** for toroidal cutters (**was 10°**); **45°** for endmill cutters (**was 10°**). This is to get better results with the different kind of cutters.

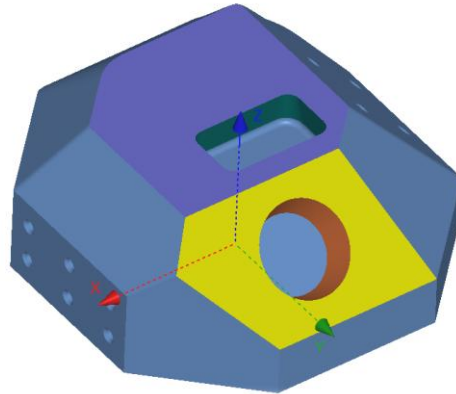
| Passes            |      | Limits                                                 |      |
|-------------------|------|--------------------------------------------------------|------|
| Passes            |      | Min                                                    | Max  |
| XY Thickness:     | 0    | Z:                                                     | 0 20 |
| Z Thickness:      | 0    | Angle:                                                 | 0 90 |
| Tolerance:        | 0.02 | <input checked="" type="checkbox"/> Contact areas only |      |
| Overthickness:    | 0    | Point Spacing                                          |      |
| Bitangency angle: | 20   | <input type="checkbox"/> Enable maximum                |      |
|                   |      | Max segment length: 0                                  |      |

**10274: Passes:** The dialog for Edit Passes to Holder now allows the Flute Length to be modified, but only if the taper angle is zero.

**10280: Five-Axis:** A new Five-Axis Deburring toolpath has been added.



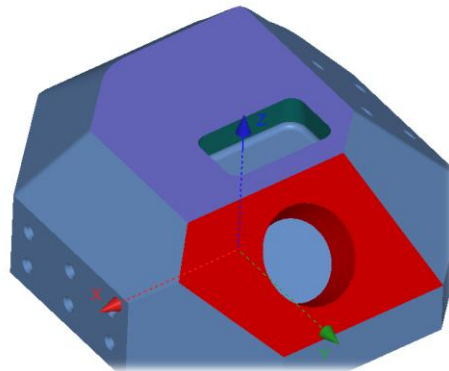
### Mini Guide to Five-Axis Deburring



For this example, we will deburr the intersection of the conical orange surface and the flat yellow surface.

It uses just the surfaces.

Generally, better results will be obtained from parts that have sharp corners.

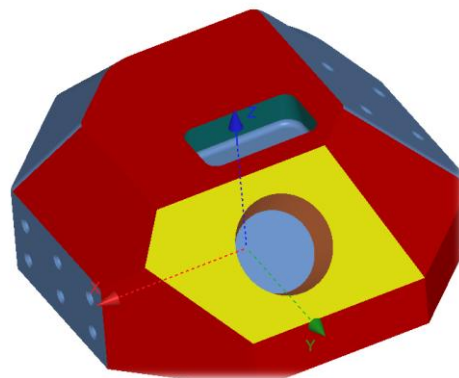


Select the yellow surface and the orange surfaces.

Then do: **Ctrl + Label geometry**. That will create the Drive triangulated surfaces.

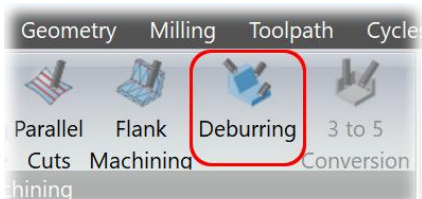
It is the surface intersections within this surfaces folder that will be deburred.

So, it will not try and deburr the intersection between the yellow and purple surfaces or blue surfaces.



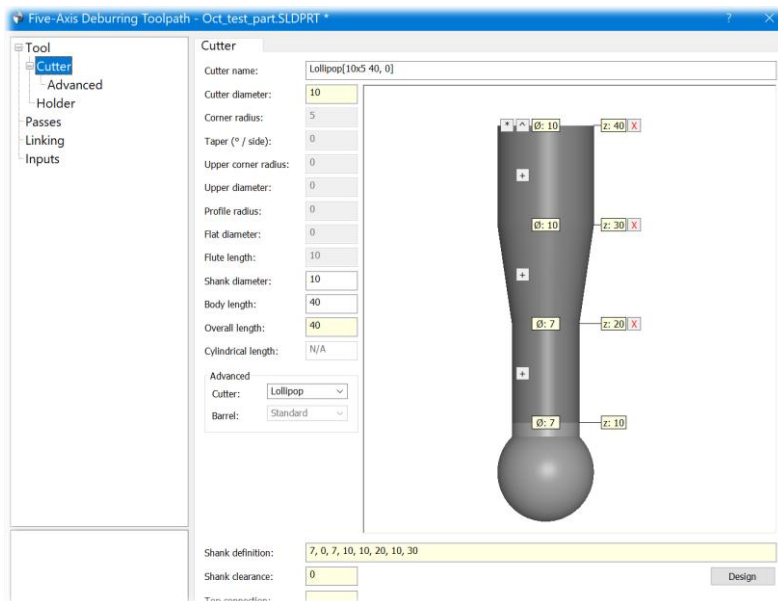
Optionally, select any surface to be protected (highlighted red in the image left).

Use **Shift + Label geometry** to create the Check surfaces triangulated folder.



Select the **Drive surfaces** triangulated folder, and then using Ctrl, select the **Check surfaces** triangulated folder (if being used).

Pick **Five-Axis Deburring**.



For this example, the following cutter was defined.

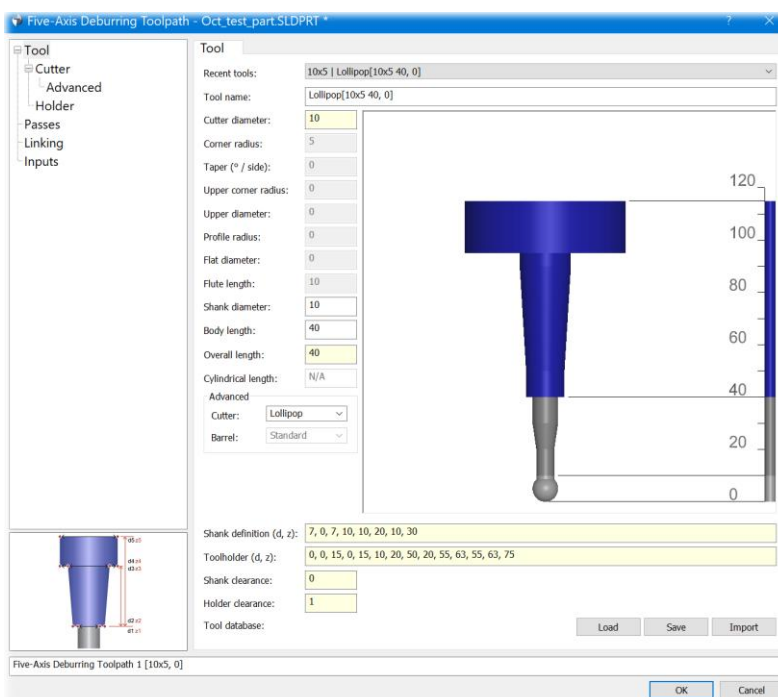
On the **Cutter** page, set the Advance cutter type, select:

### Lollipop

Cutter diameter: 10

Shank diameter: 7

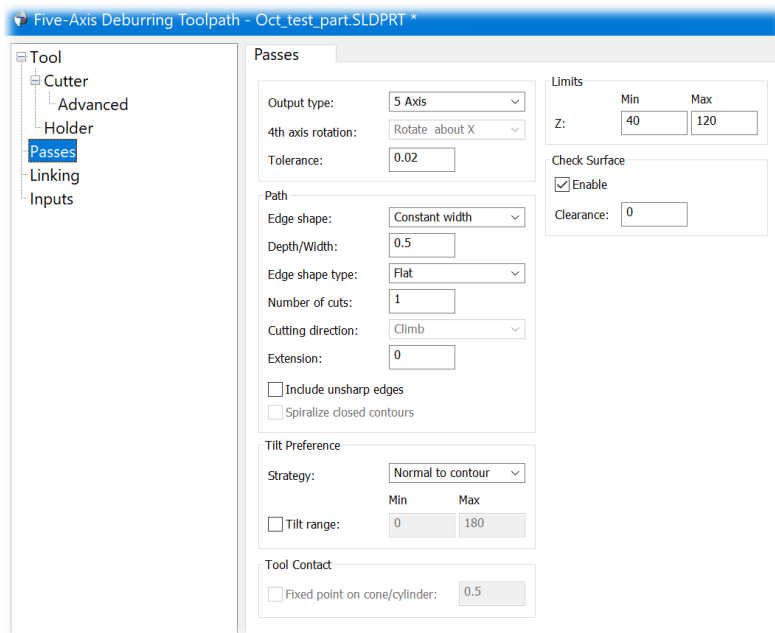
Then use the + button, Diameter and Z fields to have the 7 diameter shank to Z 20, before tapering out to 10 diameter at Z 30, and then 10 diameter to Z 40



### Define a holder

The d.z coordinates for this holder are:

0, 0, 15, 0, 15, 10, 20, 50, 20, 55, 63, 55, 63, 75



On the **Passes** page things like rotating axis can be restricted, Z limits set, tilting options as with other Five-Axis toolpaths.

Deburring passes parameters:

**Edge shape:** Allows Depth or Width to be set.

**Depth/Width:** Controls the size of the deburring, and is measured based on the **Edge shape** selection.

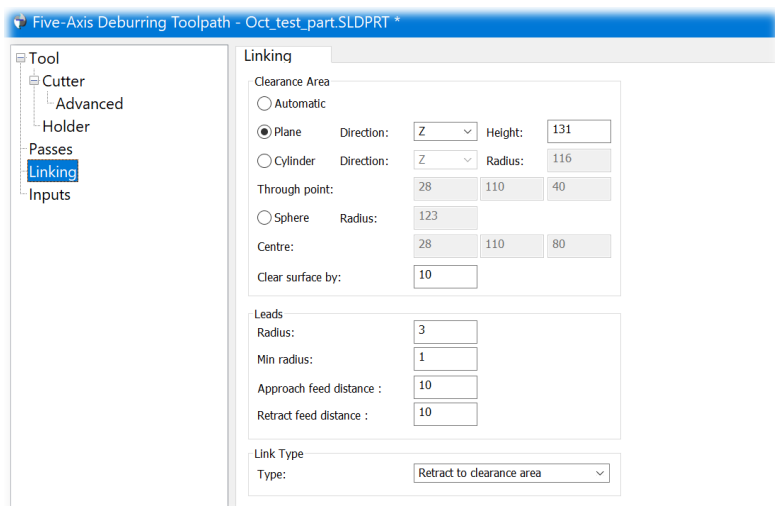
**Edge shape type:** Allows Flat or Rounded to be selected. But this is only effective if the Number of cuts is greater than 1.

**Number of cuts:** The number of passes taken to create the Flat or Rounded corner.

**Extension:** Is to allow the pass to be extended slightly.

**Include unsharp edges:** This should be enabled for parts that already have small chamfers in the CAD model, and a suitable chamfer size defined.

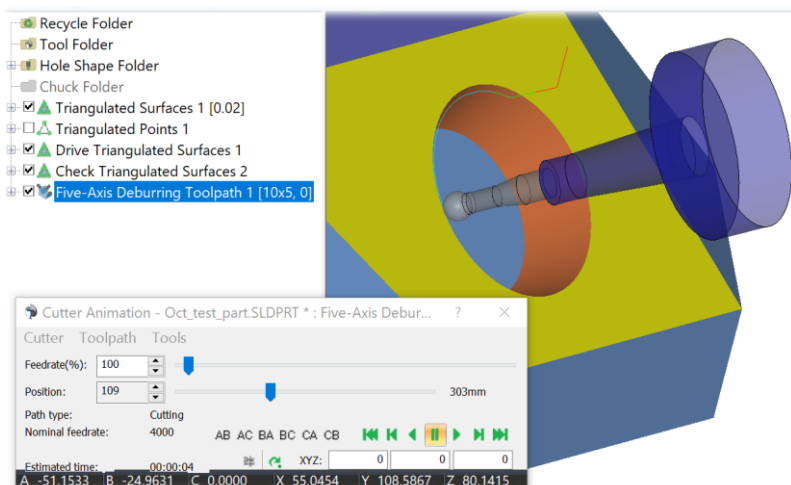
With the default Step /Width of 0.5, and then number of cuts set to 1, this will create a 0.5 chamfer.



The **Linking** page provides options for different retract types and levels.

Leave the defaults for this example.

Click OK



You should then see a toolpath running around the intersection of the orange conical surface and the flat yellow surface.

With the Five-Axis Deburring Toolpath selected, pick: **Cutter Animation**.

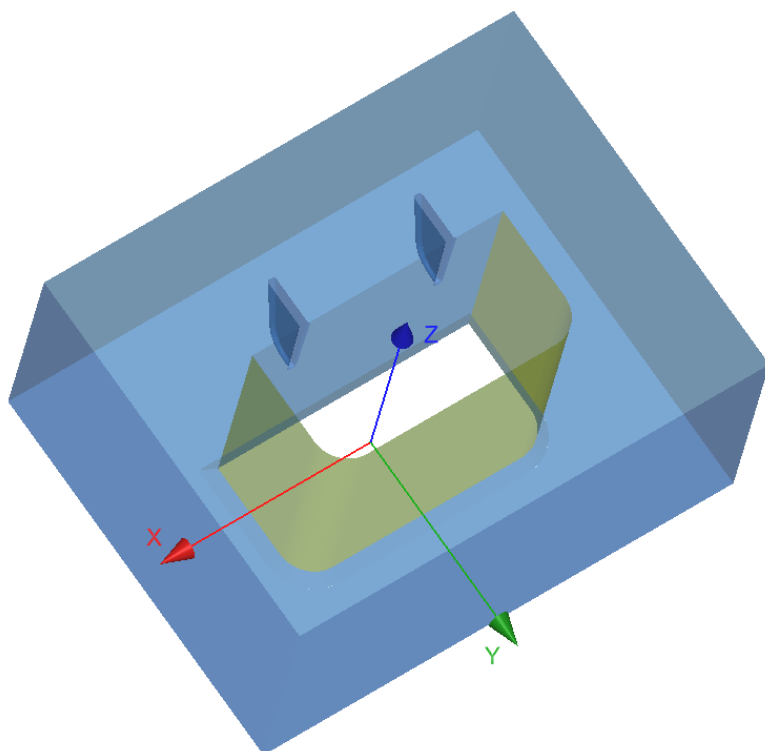
It should be like the image left.



Select the Five-Axis Deburring Toolpath, and **mouse right > Properties**.

On the Passes page, change the **Number of cuts** to: 3  
Click OK

Now there are 3 passes to create the deburring operation.



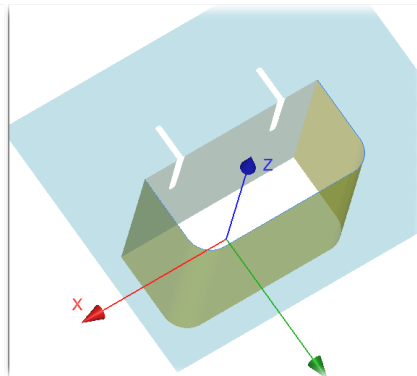
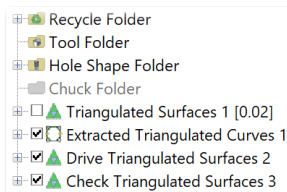
It is possible to give a curve as an input too.

That is what this example will explain.

To the left is the part in Translucent mode. Only the intersection of the top and yellow surfaces should be deburred.

The yellow surfaces were selected, and **Extract Curves** used, setting the Z min limits so only the top curve would be detected.

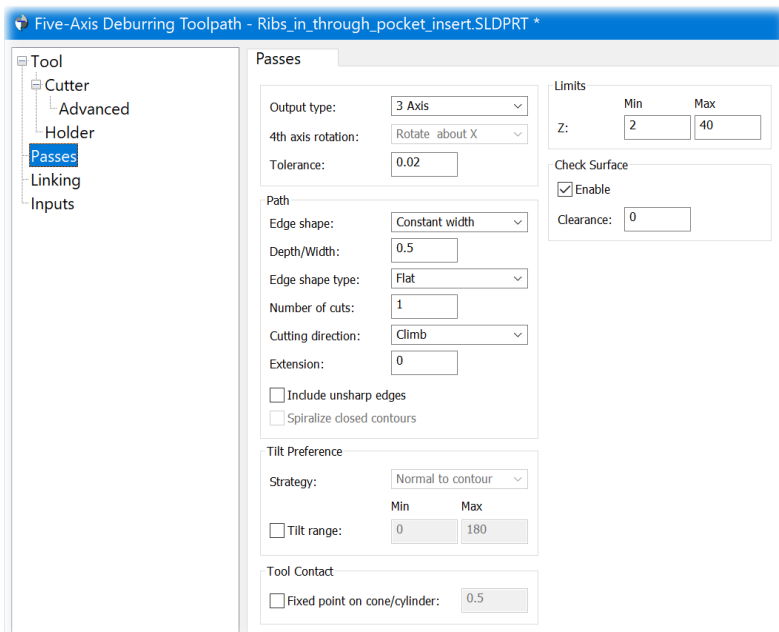
The toolpath will be set to **Three-Axis** to keep the tool axis vertical.



The yellow surfaces and the top surface were selected, and then using **Ctrl + Label geometry** separated them out as Drive surfaces.

The wall with the ribs in it was selected and using **Shift +Label geometry** separated them out as Check surfaces.

Selecting the **Drive triangulated surfaces**, **Check triangulated surfaces**, and the **Extracted triangulated curves** folders, Deburring is selected from the Five-Axis ribbon.

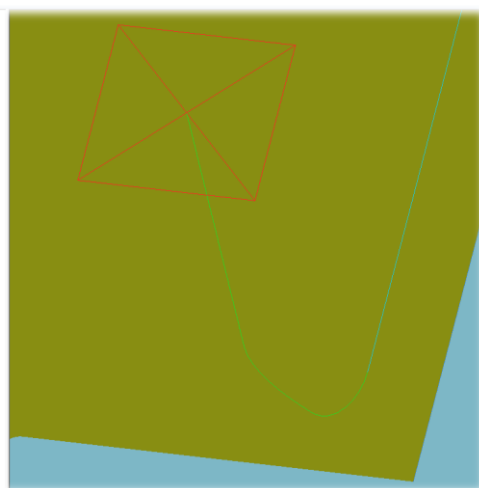
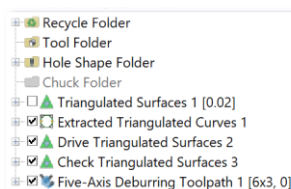


This time a ball nose cutter will be used.

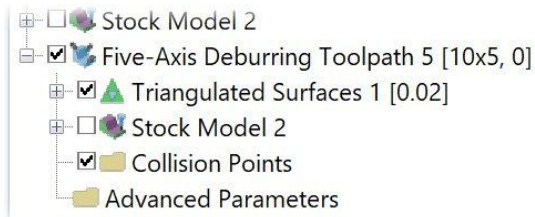
On the Passes page. Set the **Output type** to be **Three-Axis**. Everything else has been left with its default setting.

However, notice the **Cutting direction** (in the Five-Axis mode this is greyed out, unless the Fixed point on cone / cylinder' option is enabled.), but if constrained to Three-Axis, it is possible to set **Climb** (the default) or conventional milling if required.

The default linking parameters were used.



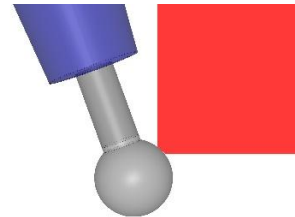
Looking at the toolpath here, it can be seen to arc down and round into the cutting. It only deburs what can be accessed around the curve without touching the surfaces in the Check surfaces folder.



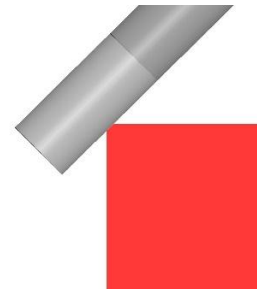
It is also possible to provide a closed stock model as an input, so the Deburring toolpath is only calculated inside the supplied stock.

### Deburring - Tool types supported.

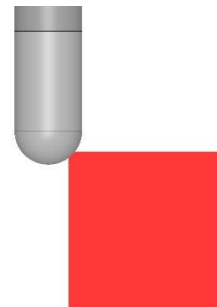
Using a deburring tool like a lollipop cutter may in some cases give better access to some areas.



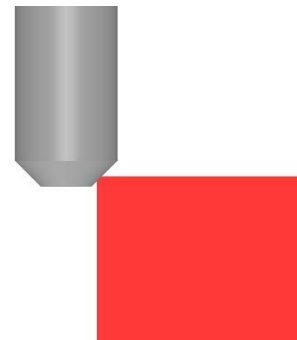
Using the side of a cylindrical cutter, useful for creating a small chamfer.



Using the tip of a ball nose cutter. With a ball nose cutter also the side of the flute can be used, when **Fixed point on cone/cylinder** is ticked.

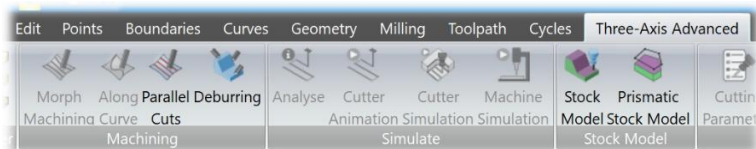


Using a chamfer cutter



**10531: Three-Axis Advanced:** A new Three-Axis Advanced optional module has been added to allow to do deburring in Three-Axis. And some other Five-Axis style toolpath's to be used while locked to a Three-Axis mode. It will allow small under cuts / back tapers to be machined using Lollipop, Teeslot, Dove tail, Barrel, Lens style cutters on a Three-Axis machine. Above is explained how to do the deburring, which is basically the same in the Three-Axis Advanced module. Below is a Mini Guide how to create a Three-Axis toolpath to machine surfaces with an undercut.

**Note:** This is a licensed module option. You would need to speak to your reseller about price and to get this option enabled and a new license code. These toolpaths can be posted with the exiting macro posts.



### Mini Guide to Advanced Three-Axis toolpaths.

Using the Advanced Three-Axis toolpath's are different to how you would create waterline or raster passes for example.

Firstly, you need to consider the type / format of the part file that is being opened. If the part is an igs file, it is important that you convert it to NURB's, failing to do so will result in the Advanced Three-Axis not being calculated.

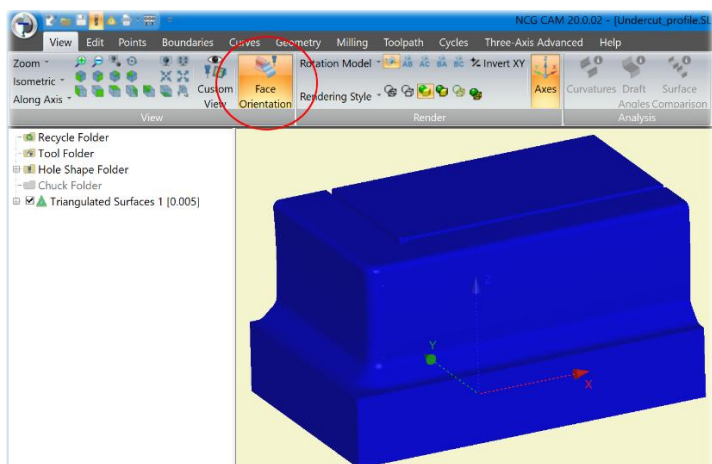
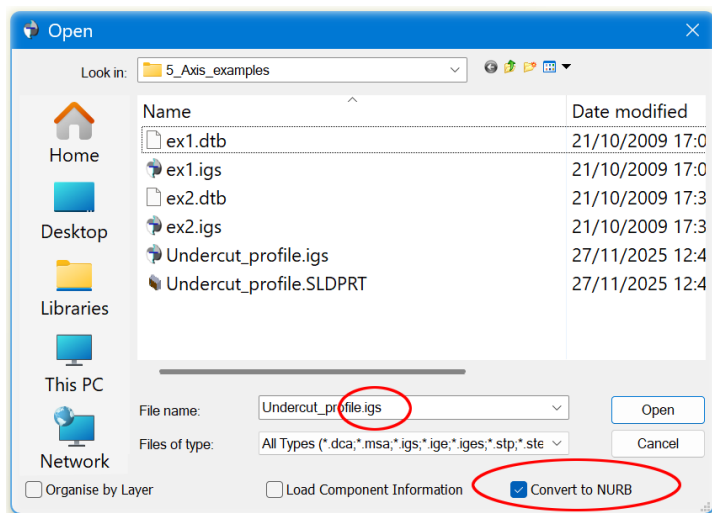
Step, Solidworks, Parasolids, Creo, Catia, UG, and Rhino parts will be imported as NURB's. by default.

Another difference is that you create a toolpath directly. It's not passes and then linking to create a toolpath.

The surfaces Face Orientation is important, and needs to be checked, as it controls which side of the surface the cutter will cut.

In the View ribbon, click on the Face Orientation. This will render the surface in different colours (front and back – blue and red by default but can be set in Options > System colours.

For the image to the right, the surfaces Face Orientation is all the same way and the front face shown.

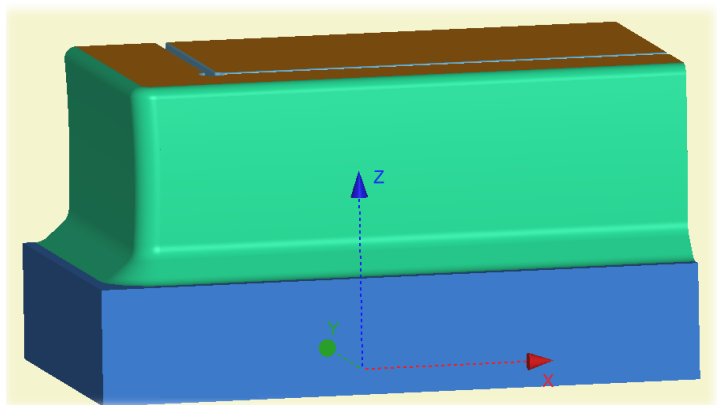
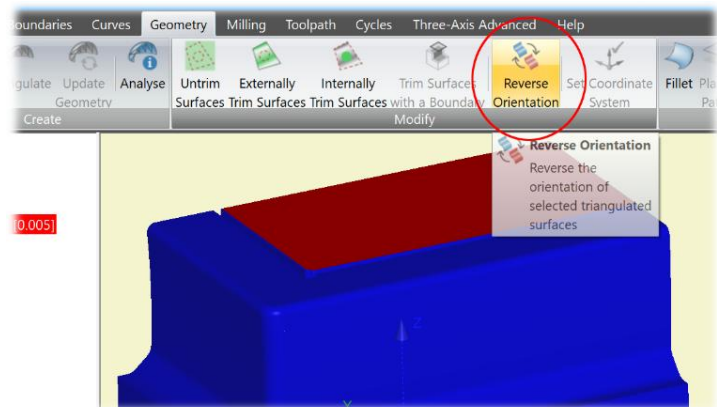


The surfaces, Face Orientation can be the wrong way for all or some of the surfaces. In those case's the surface where the Back is shown, those surfaces need reversing so the Front is shown.

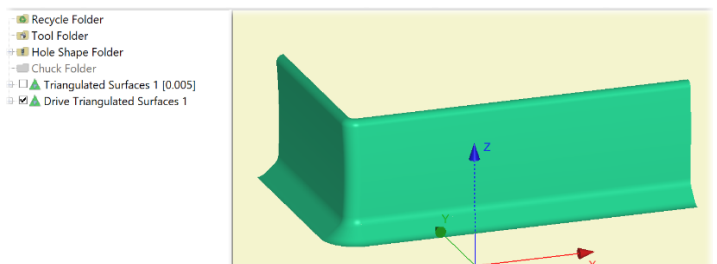
Select the surfaces that need to be reversed (remember sometimes selecting what you don't want and inverting the selection can be quicker)  
Then, from the Geometry ribbon, click Reverse Orientation.

Once sorted, click on the Face Orientation button in the View ribbon, that will show the surface colours again.

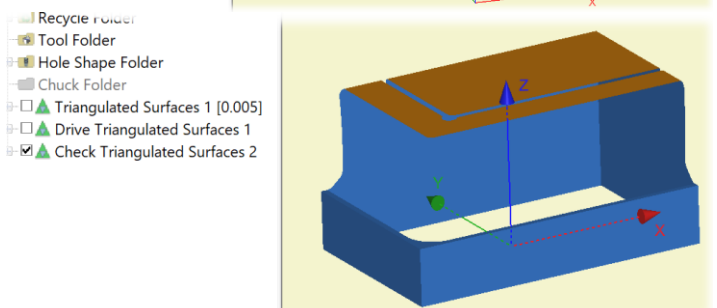
The surfaces to be machined (Drive surfaces) and the surfaces to be protected (Check surfaces) need to be separated into different folders,  
For this example, the green surfaces are to be machined to be able to machine an undercut form.



Select the surfaces to be machined (the green surfaces here).  
Right click, and then Ctrl + Label Geometry. (you can also do a Ctrl + Label Geometry from, the Edit ribbon).  
That will create a new folder: Drive Triangulated Surfaces. (Shown right)



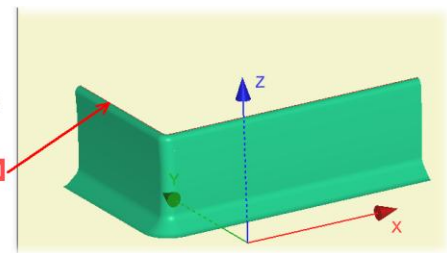
Select the surfaces to be used as Check surfaces, these are surfaces you don't want the cutter (or holder) to touch but can go up to.  
Right click and then Shift + Label Geometry. (you can also do a Shift + Label Geometry from, the Edit ribbon).  
That will create a new folder: Check Triangulated Surfaces. (Shown right)  
It is possible to have and use two Check Triangulated Surfaces.



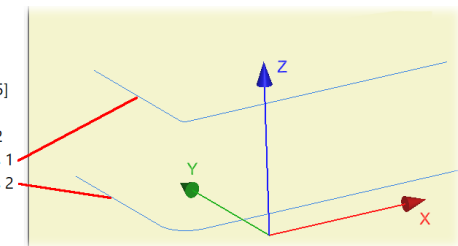
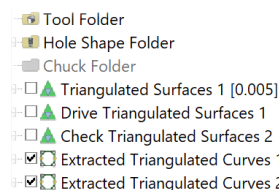
**Note:** The use of Ctrl and Shift has just added the Drive / Check prefix to the folder name, it has not modified the surfaces. It just helps identify them in the contents tree.

For this example, the plan is to use a Morph Toolpath, morphing between curves to machine the green surfaces. The task will be to extract the curves needed, in this case one around the top of the Drive Surfaces, and one around the bottom edge of the Drive Surfaces.

Select the Drive Triangulated Surfaces folder, then from the Curves ribbon, pick Extract Curves. As the top and bottom edges of the drive surfaces is horizontal, the Z min and max In the Extract Curves dialog can be used to help restrict where curves are extracted from. By changing the Z Min to be just below the Z Max, only the top edge of the surfaces will be considered for curve extraction.



Extracting the curve at the bottom of the Drive surfaces is just the same as above, but the Z Max is adjusted to be just above the Z min.



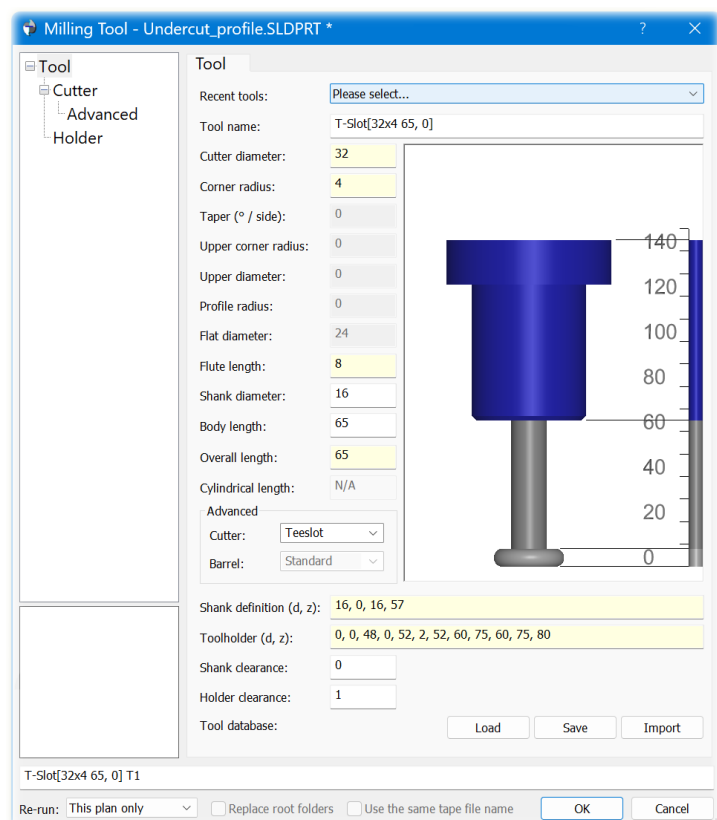
The same curves could be extracted from the Check Surfaces, but would have been more work, deleting unwanted curves and joining the wanted bits – use whichever is easier.

The plan is to use a Teeslot style cutter with a full radius so the undercut areas can be accessed.

Define the cutter and holder in the Tool Folder. Mouse right on the Tool Folder in the contents tree and pick Create Milling Tool, that will display the Tool dialog.

Firstly, select Teeslot in the Advanced > Cutter area – this will change some of the parameter values and which parameters are available for the selected cutter type. Then the Cutter diameter, Corner radius, Flute length, Shank diameter, and Body length can be set. A suitable Holder can also be added.

**Note:** The tool could also be defined as the toolpath parameters are set or loaded from a tool database as the toolpath is created.



Select the Drive Triangulated Surfaces and then the Check Triangulated Surfaces. Select the Top Curve, and then the Bottom Curve, and then Tool if defined the Tool Folder.

The order is important, surface wise the first surfaces folder selected are the surfaces to be machined, the second (and sometimes third) surfaces folder selected are the surfaces to protect against. It is similar for the curves, the passes will be created from the first curve to the second curve. Selecting the tool first or last would not make any difference.

Then from the Three-Axis Advanced ribbon, pick Morph Machining.

Go to the Passes page. The Thickness can be set for XY and for Z. Under Path > Cutting pattern, select Morph between curves.

Cutting type could be Oneway, Bidirectional or Spiral (which is useful for a closed form). Oneway or Bidirectional would be suitable of an open part like this example.

The stepover can be set.

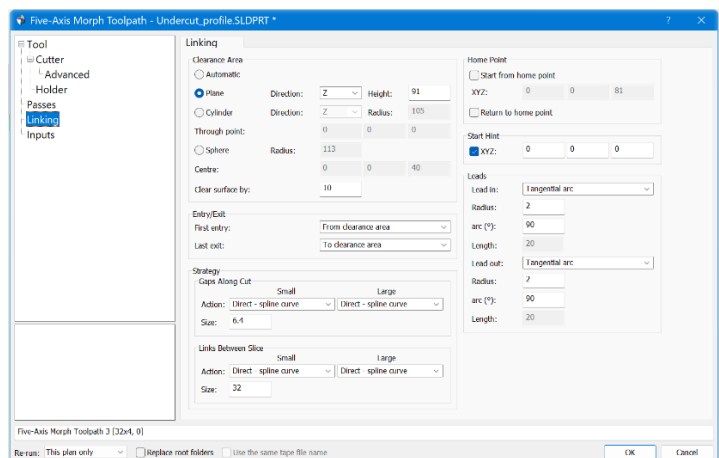
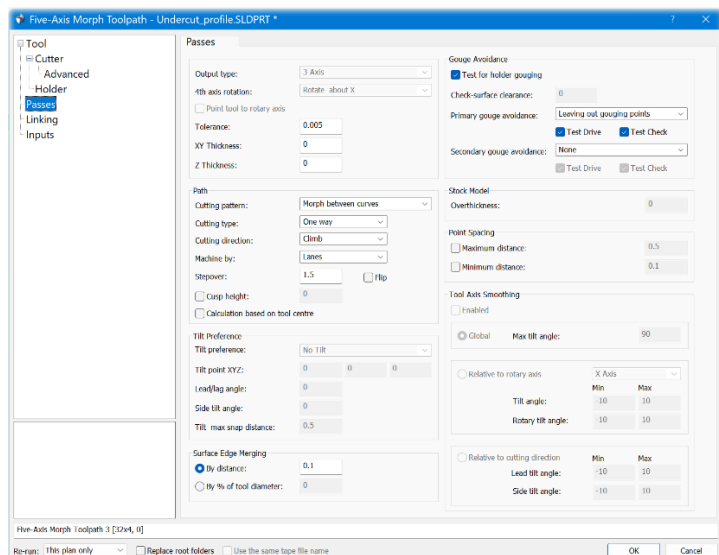
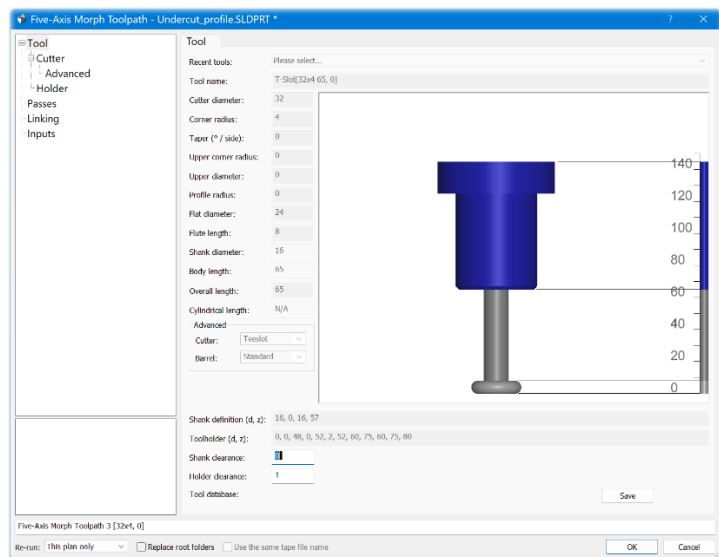
If the curves had been selected in the wrong order, Flip can be used to reverse the ordering.

A lot of options are greyed out and will remain so, as this dialog is a restricted or limited Five-Axis dialog

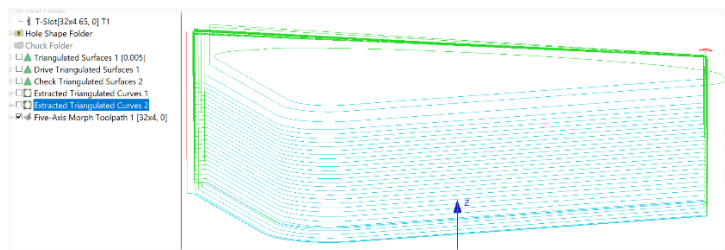
Go to the Linking page.

For this example there is no need to change anything, but the options are there.

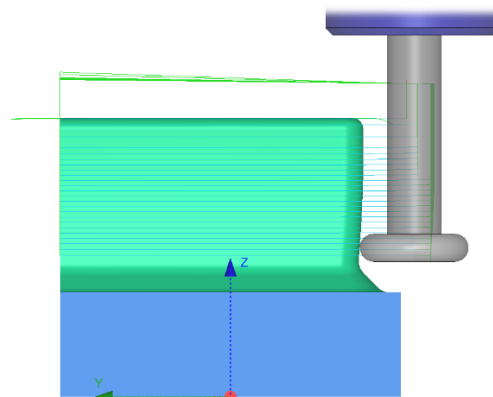
For this example nothing needed to be changed.



The resulting toolpath.



Animating the toolpath showing the undercut being respected.



Advance Three-Axis toolpaths available:

Morph – With options for Between curves, To a curve, Between surfaces, To a surface.  
 Along curve – With Projected or Along options.  
 Parallel cuts.  
 Deburring – With Flat and Rounded options.

**10065: Area Clearance Passes:** The default tolerance for creating area clearance passes is now limited to a maximum of 0.3 (it was 0.5), to avoid the passes becoming too coarse.

**10378: Pencil Passes:** A problem has been fixed where a non-endmill cutter could gouge when a substantial **Overthickness** is applied.

**9896: Pencil Passes:** A problem has been fixed where there was the possibility of the toolholder gouging the part, if the cutter had an **Overthickness** applied.

**10346: Rest Roughing:** Rest Roughing and Rest Horizontal Passes are now greyed out in the User Interface if unlinked machining passes or drilling cycles are supplied as inputs.

**10085: Selected Surface Machining:** The internal underlying Bounding box used for the passes calculation has been improved, giving performance improvements especially for 3+2 machining on bigger parts.

**10174: Rest Finishing:** The performance has been improved when using a toroidal cutter for the passes being created.

**10419, 10503: Along Curve Passes:** Fixed a problem, where offset Along curve passes could be created as open curve when the curve is closed (or looped).

**10049: Pencil Passes:** A problem has been fixed where passes were not always being created within small features of some parts.

**10337: Pencil Passes:** Eliminated gaps that could occur in pencil passes where an **Overthickness** has been defined, particularly for endmills.

**10222: Pencil Passes:** A problem has been fixed where gaps in multiple fragments of pencil passes around upstands rising from sloping surfaces can occur.

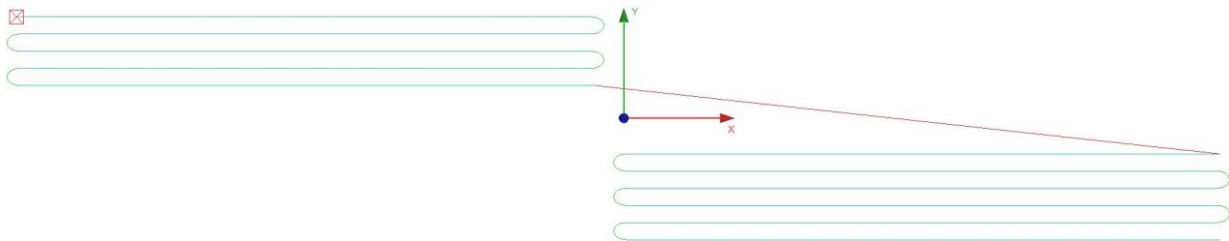
**10294: Pencil Passes:** Some gaps and overlaps of a small number of microns have been eliminated; this can reduce the number of linking moves. This partly fixes tickets 10293 and 9386 also.

**10306: Pencil Passes:** A problem has been fixed that could lead to gaps of a few microns when a value other than 0 has been defined for the **Overthickness**.

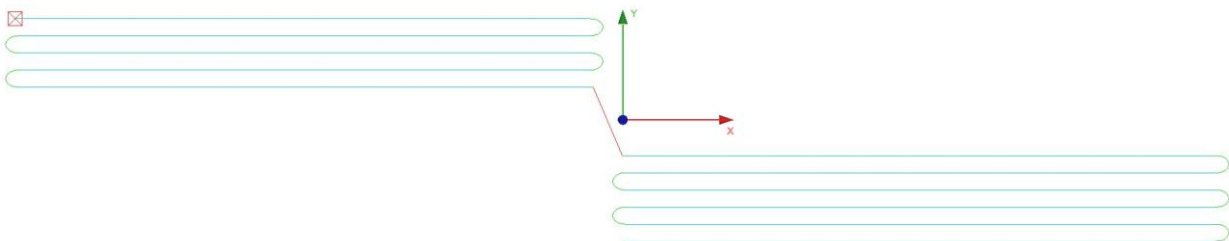
### Toolpath Linking:

**1857: Linking:** The bidirectional linking of Boundary, Raster, Radial, Spiral passes now uses the flexible "reverse as best" approach as used for UV passes and open Z slice style passes, rather than the original rigid reverse by adjacency approach. This should lead to generally shorter bidirectional toolpaths.

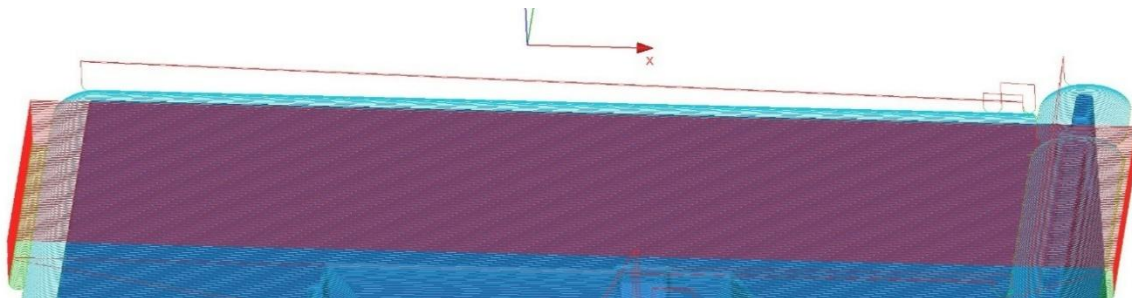
In the following example two areas are machined. In the second area, the first pass runs in the opposite direction to the last pass of the first area, since the direction is always reversed.



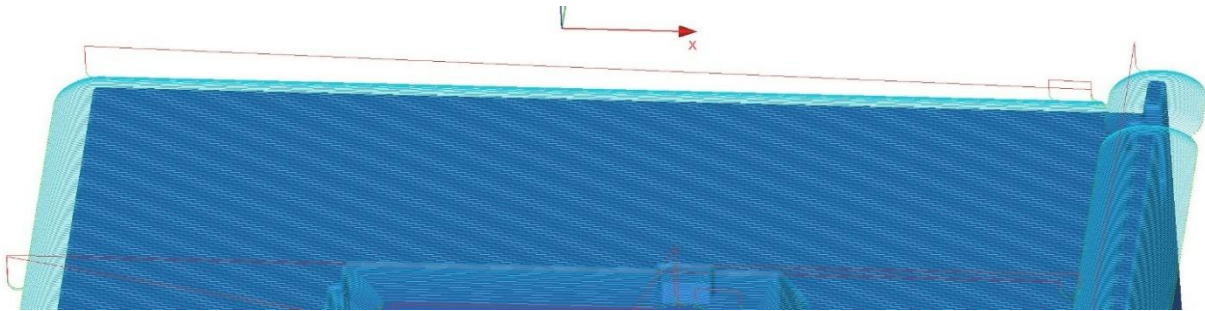
If reversing the pass direction can shorten the rapid positioning move, the direction is now adjusted automatically. This means that the first path of the second area and the last path of the first area run in the same direction, significantly reducing the length of the rapid move between the areas.



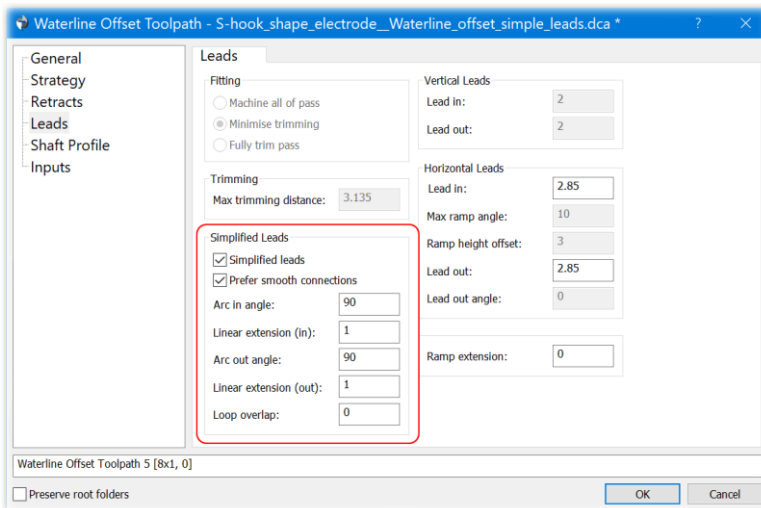
In the following example, the raster passes are split into pieces by an angle limitation. The consistent change in direction between the passes results in lots of rapid moves.



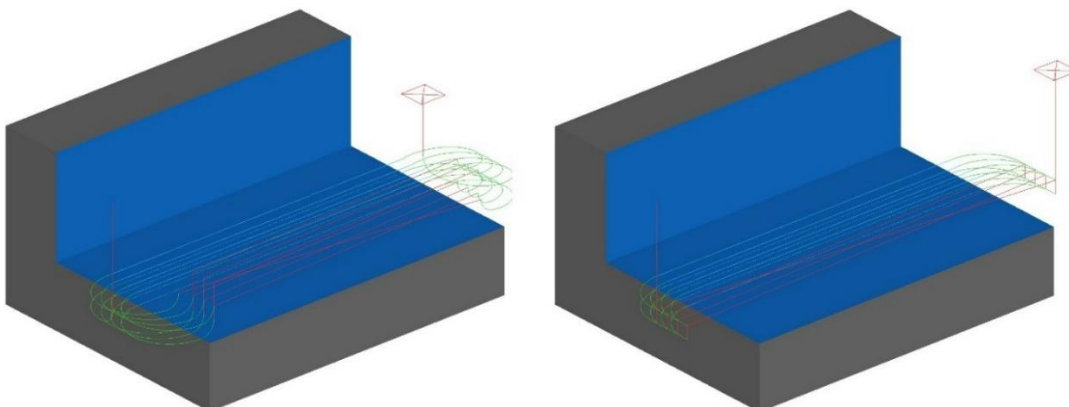
In v20 the passes are intelligently linked using identical settings. This method eliminates almost all unnecessary rapid moves.



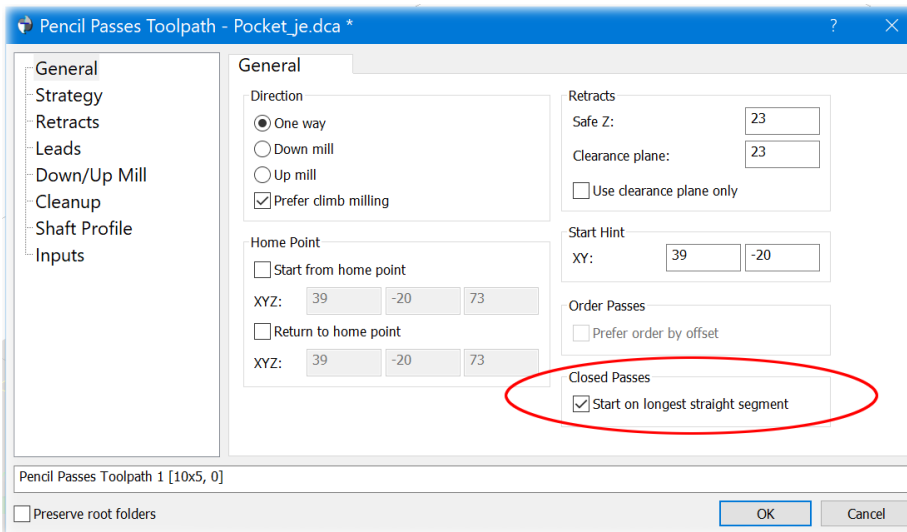
**10307: Linking:** It is now possible to use the Simplified Leads options from Waterline Toolpaths for Waterline Offset Toolpaths



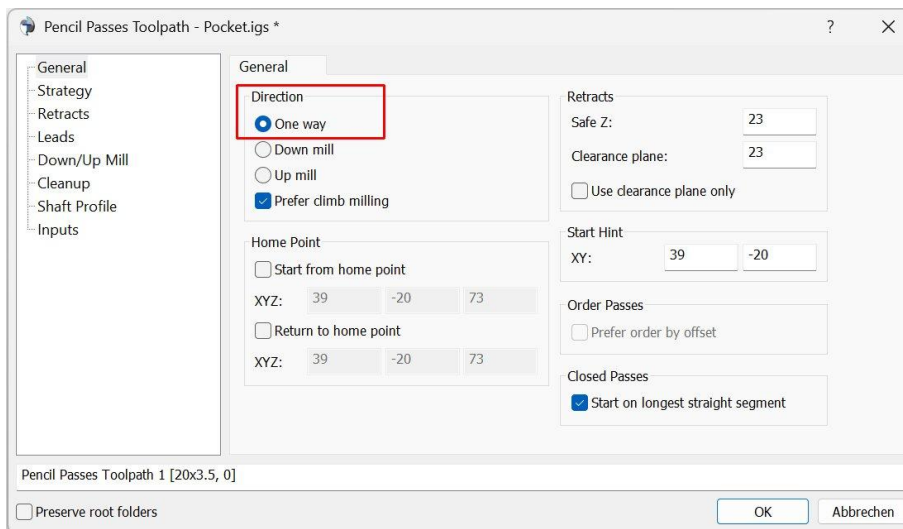
The following example shows a Waterline offset toopath. On the left are default horizontal leads, on the right Simplified Leads.



**10282: Linking:** An option has been added to Pencil Passes Linking to link to the longest straight section of a closed pass, (as already exists for Machine Along Curve linking), then option is enabled by default.



**10287: Linking:** The default linking direction for pencil passes has been changed from **Down mill** to **One way**, as this is more often used.

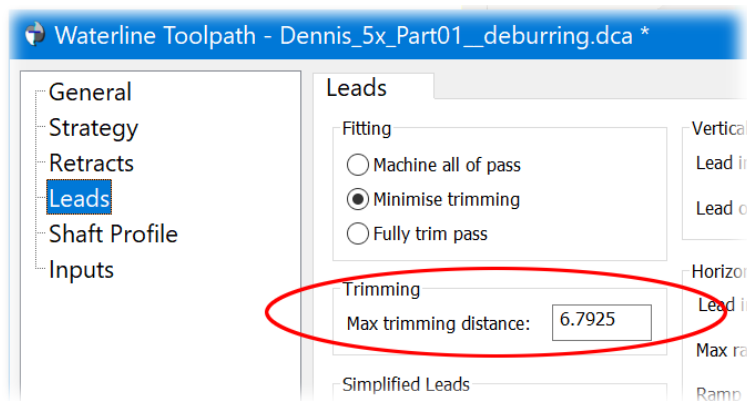


**10288: Linking:** For pencil passes, the default value of the **Minimum pass length** for endmill cutters has been increased from 0.02 to 0.25.

**10448: Linking:** Fixed a problem with the fitting of Horizontal lead arcs which could lead to empty or invalid fragments and disjointed toolpaths in rare cases.

**10267: Linking:** Previously it was possible for a ZigZag roughing toolpath to plunge into the first path when a ramping move could be applied. This has been fixed.

**9986: Linking:** The default **Max trimming distance** for leads for Waterline, Waterline Offset, Waterline Stepover, and Helical passes now consider the Horizontal lead arc radii as well as the Vertical lead arc radii. This should reduce the possibility of retracting moves being made against a vertical wall.



**10544: Linking:** In the case where the start and end of lead in and lead out moves were directly above each other in Z we were unnecessarily retracting to Z Clearance if Full Vertical retracts had been selected. This has been fixed.

**10097: Linking:** Fixed a problem which could lead to unnecessarily high retracts in some circumstances.

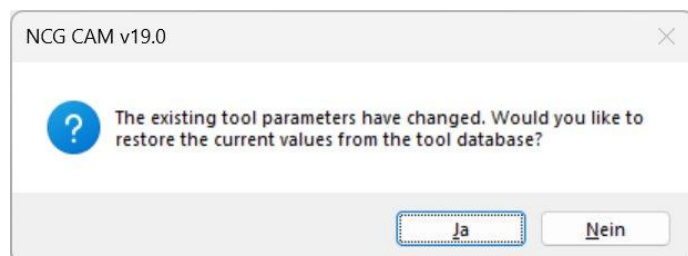
**10145: Linking:** Fixed a problem when linking Core roughing passes which could result in some retracts not respecting the clear by distances.

**10172: Linking:** Fixed a problem which meant the top slice of a helical passe could be machined after the lower passes.

**10535: Linking:** At along curve passes we were unnecessarily adding retract moves in cases where the end points of passes (curves) meet. This has been fixed.

### Tool Database:

**10122, 10133: Tool Database:** There was a problem where the plan dialogs were giving a 'Parameters modified' pop-up message for a tool which was loaded from the tool database when it had not been modified, this has now been fixed.



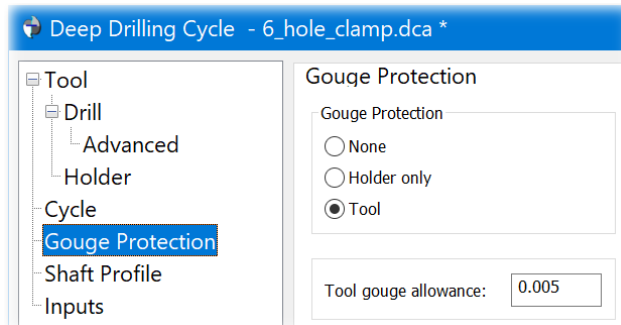
### Detect Holes:

**10380: Detect Holes:** The defaults for detecting holes have been modified to improve the detection of holes in cylindrical cases. The Minimum hole diameter has increased from 0.05 to 0.5 on the Dialogs Filter page. This also fixes ticket 10398

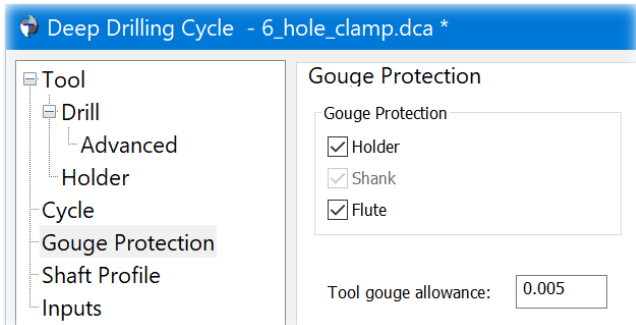
**10161: Detect Holes:** There was a problem when selecting UV Direction surface folder as plan input to Detect Hole Features failing, this has now been fixed.

## Drilling:

**9637: Drilling:** Separate gouge protection options are now available for the **Flute**, **Shank** and **Holder**. If the gouge protection is enabled for the flutes, then it is automatically enabled for the shank.



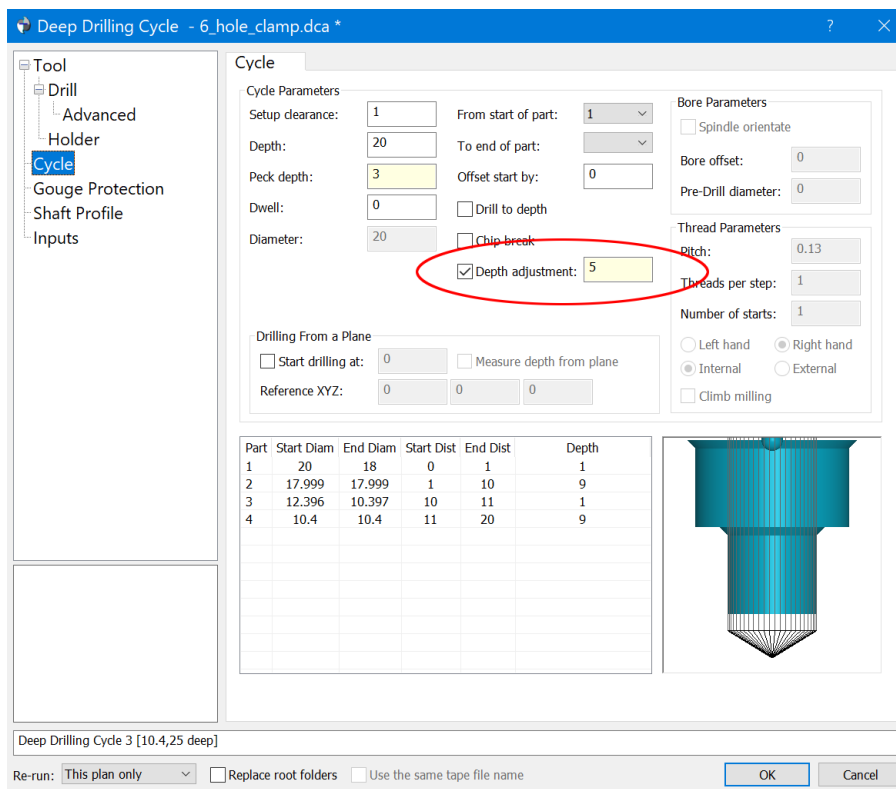
In v19.0.15 and earlier



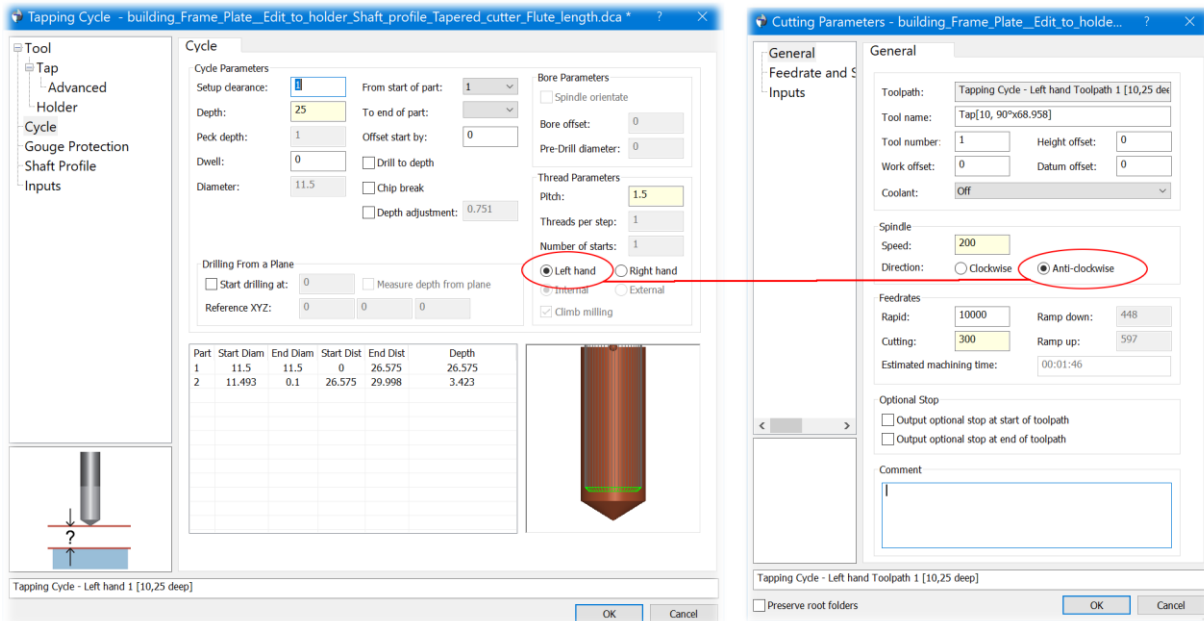
In v20.0.01

**10359: Drilling:** Drilling to depth gouges are now output as a DANGER message as other gouges would be.

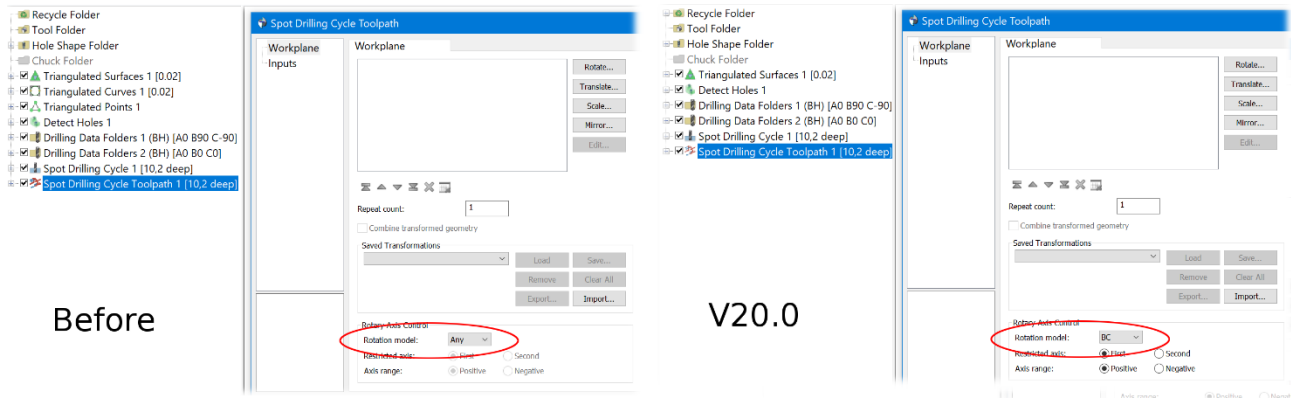
**10010: Drilling:** The depth adjustment parameter for drilling cycles has been moved from the tool page to the cycles page as it is more cycle related than part of the tool definition.



**10255: Tapping:** We now consider tapping direction when setting spindle direction in cutting parameters plans, automatically setting the correct spindle direction.



**10080: Drilling: Transformations:** We now pick up the preferred rotation model when drilling, for consistency with the milling operations.



**10297: Hole Shape Database:** When adding a new hole shape and changing the Z-ranges, the graphics were not updated correctly. This issue has been resolved.

**7546: Bore Milling:** The default peck depth is now set from the Tool Stepdown, set in the tool database, or on the cutters Advance page.

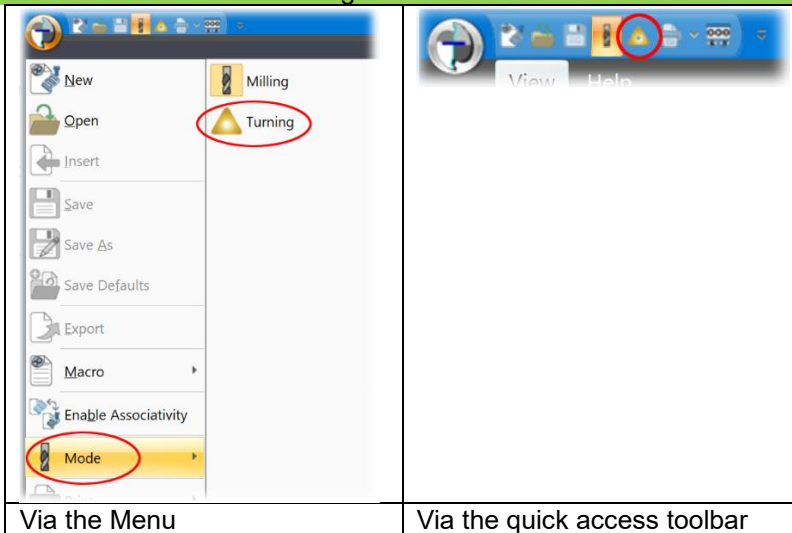
## Turning

**8981: Turning:** NCG CAM v19.0 introduced the initial turning for external profile, NCG CAM v20.0 builds on that, adding internal turning processes.: Roughing, Profiling, Grooving (radial).

The turning ribbon, and mouse right context sensitive menu has External and Internal turning passes options.



### Mini Guide for Internal turning



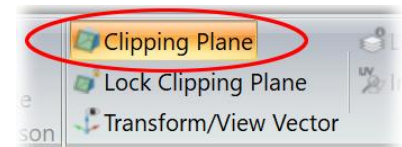
NCG CAM will always open in the **Milling mode**.

For any turning, that needs to be changed. To use the turning features, the **Turning mode** must be selected.

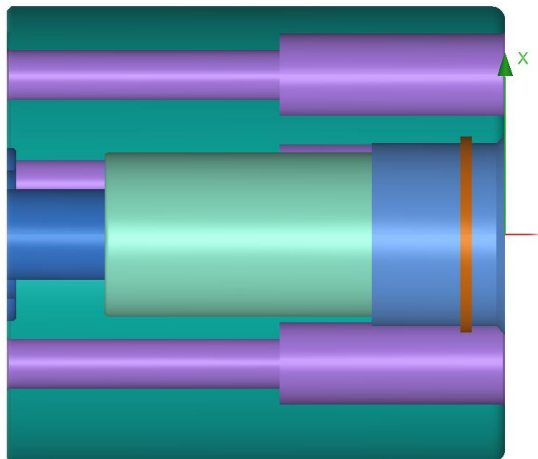
This can be done from the **Quick access** toolbar, or from the **NCG CAM menu**.

The change can be made either before or after opening the part. You can switch between Milling and Turning modes at any time.

For a part requiring internal turning, enabling the **Clipping plane** will make it easier to see what you have and what you are doing. From the **View ribbon**, click on the **Clipping plane** button.

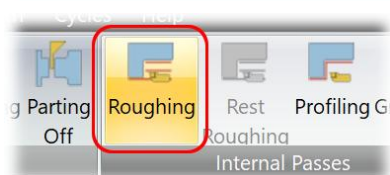


The example part, with the clipping plane enabled.

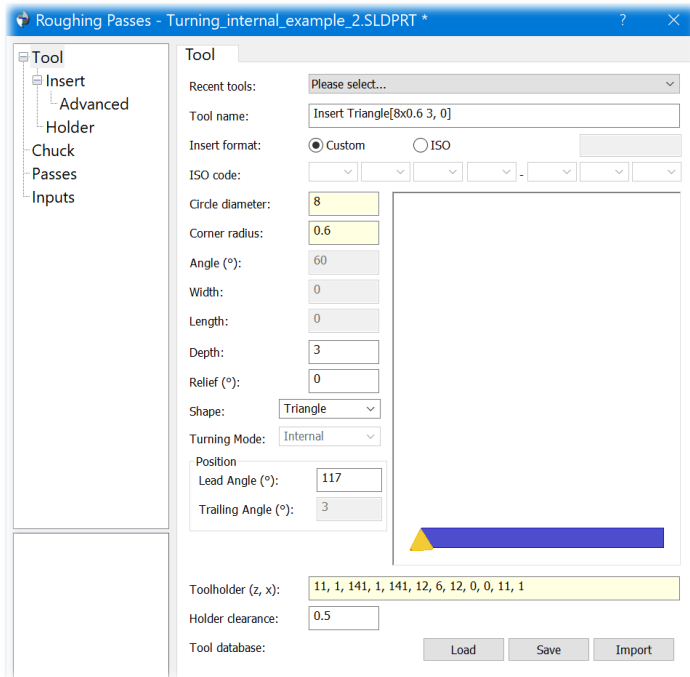


While not documented here, a 20.0 diameter hole has been drilled through the part.

The following steps: Rough out the remaining material in the bore, profile the bore to size. And then create the orange groove detail.



Select the Triangulated surfaces, and then from the Turning ribbon, pick **Roughing** from the Internal passes section.

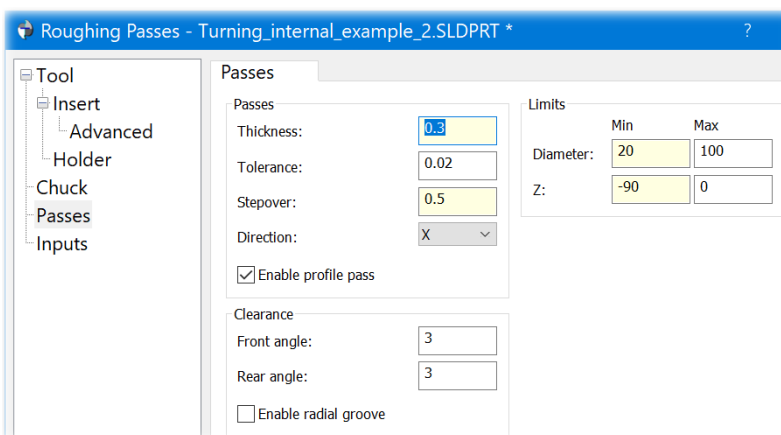


Having selected an internal turning operation, the insert is positioned accordingly.

When creating the tool holder, create it as it would be for the turning (along the Z axis).

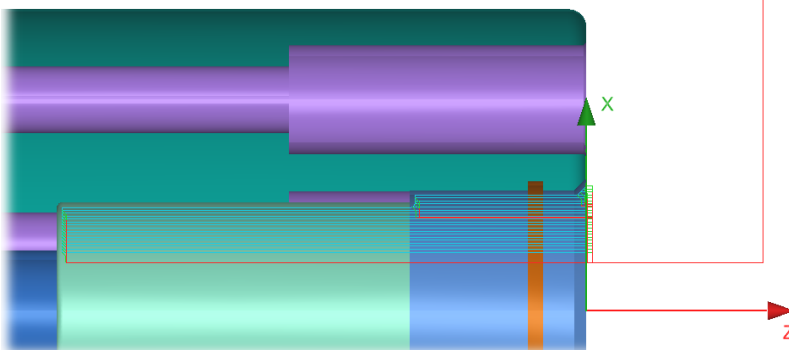
Optionally the tool could have been defined in the Tool folder, and included in the selection before picking the turning passes.

If a chuck had been defined that could also be included in the same manner.



On the passes page a suitable **Thickness** and **Stepover** can be set.

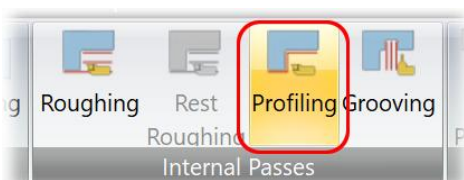
For the **limits**, the **X Min** is set to the size of the pre-drilled hole, **Z Min** has also been set to the desired turning depth – so it can't attempt a pass in the 20.0 diameter through hole.



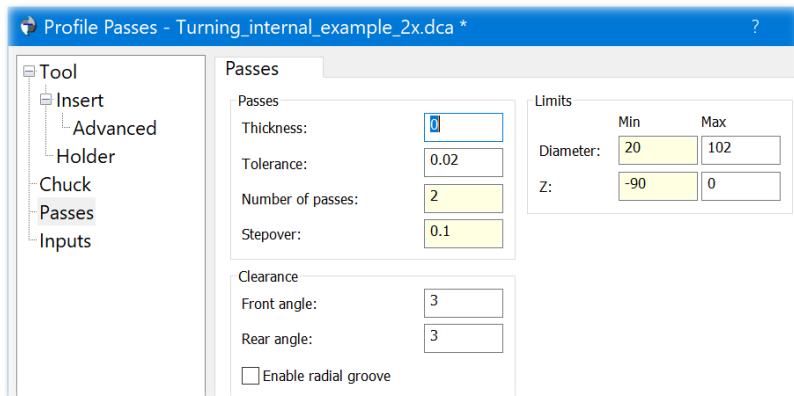
The **linking** is the same as linking external roughing passes.

For the toolpath to the left, the Home point option was used.

This has the bore roughed out to the 0.3 thickness set in the passes.



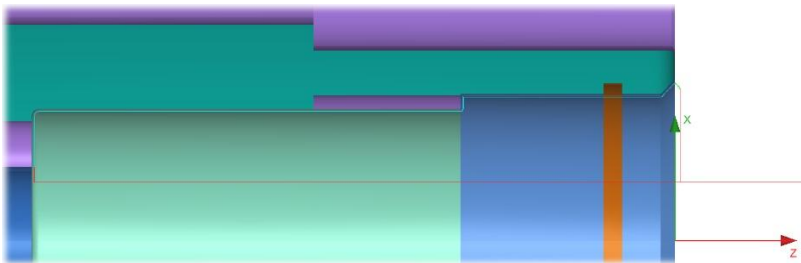
Select the Triangulated surfaces, and then from the Turning ribbon, pick **Profiling** from the Internal passes section.



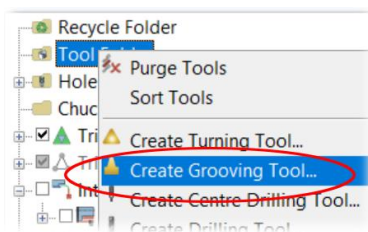
Select the previous tool from the reset tools list, but change the corner radius to 0.2.

On the Passes page, as with the roughing, the **Min dia** and **Min Z** have been modified to ensure there is no turning in the 20.0 diameter through hole.

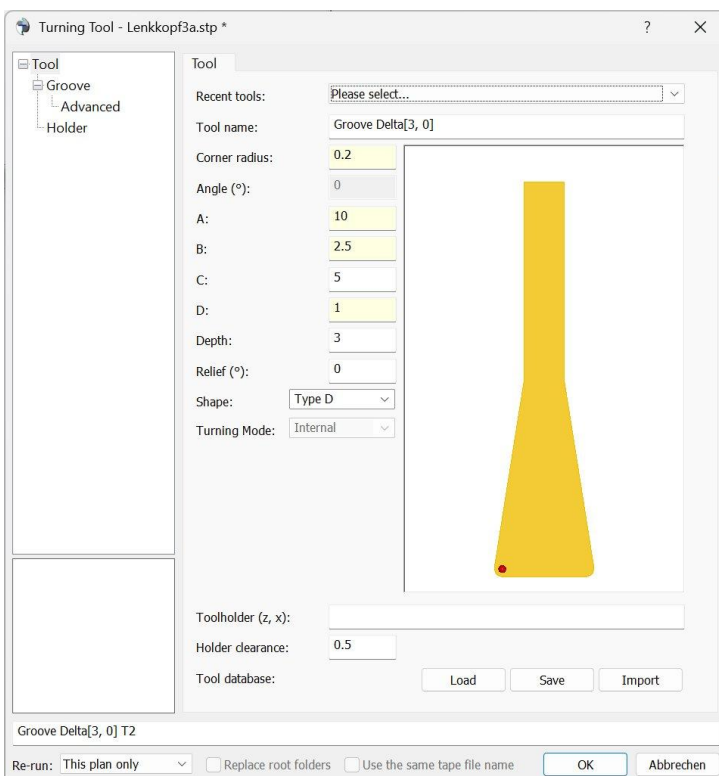
The **Number of passes** has been set to 2, and a 0.1 Stepover is set. Now 2 passes with a 0.1 depth of cut will be made, to remove the 0.3 of stock left when roughing.



The passes are **linked** to make the toolpath.



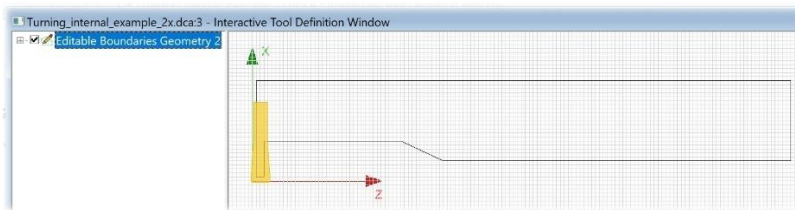
Creating a **Grooving tool**: Click Mouse right over then **Tool Folder**, and pick **Create Grooving Tool**.



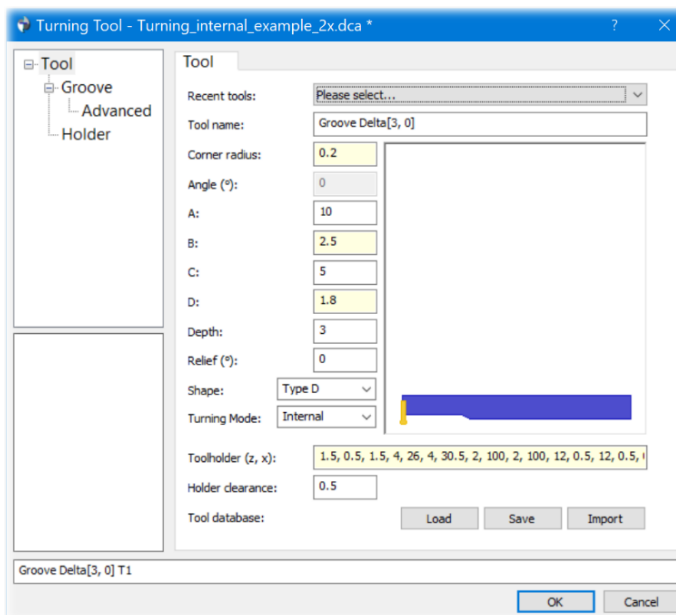
Initially, selecting the insert Shape, in this example **Type D**. and the tools Turning mode as **Internal**, as these will affect some of the other parameters default settings. (Setting other parameters, then setting the insert Type or Turning mode could change parameters you had just set).

Then set other parameters to define the inserts corner radius, widths and lengths.

Optionally, you could go the Advance page and set things like the tool number, cutting speed, feeds and coolant.



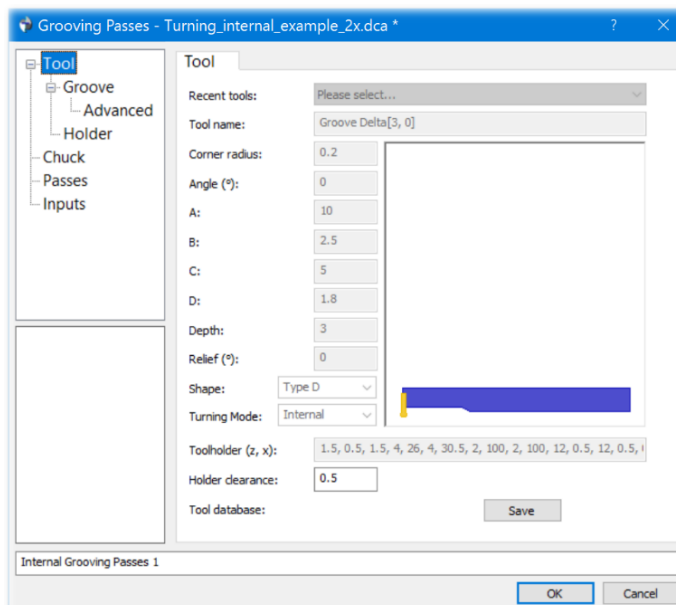
Defining the **tool holder**. Using the **Design** option is the easiest way to define a tool holder for turning. Click the Design button, then in the Design window, make mouse left click to draw the tool holder. Using the Grid and Grid snapping helps.



Click the **Import to Definition** button to import the holder shape, then the Design window can be closed.

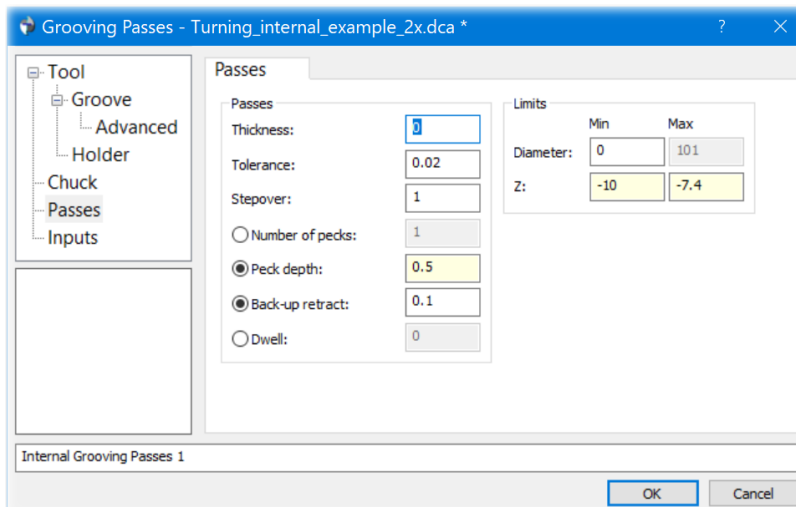
Optionally, you may want to change the **Tool name**.

Clicking OK will close the window and add this tool to the Tool Folder.



Select the **Triangulated surfaces** folder and the **grooving tool** from the **Tool Folder**. Then from the Turning ribbon, pick **Grooving** from the Internal Passes section.

The grooving dialog will have the parameters populated from the selected tool, but are greyed out. This is because those details are properties of the tool that was given as an input to the passes about to be created.



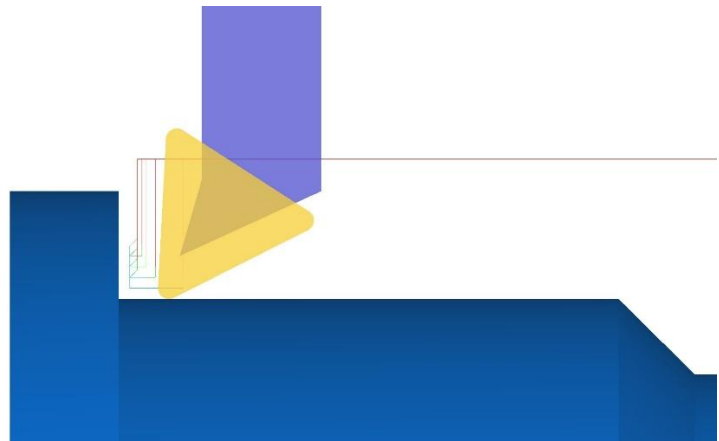
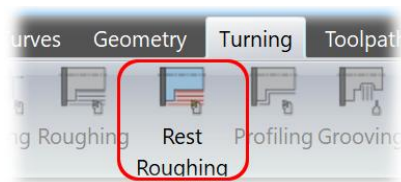
On the passes page, parameters need setting for the **Stepover**, **Peck depth** (or **number of pecks**) and any retract or dwell values.

The **Min Diameter** can be set to the diameter the grooving needs to start.

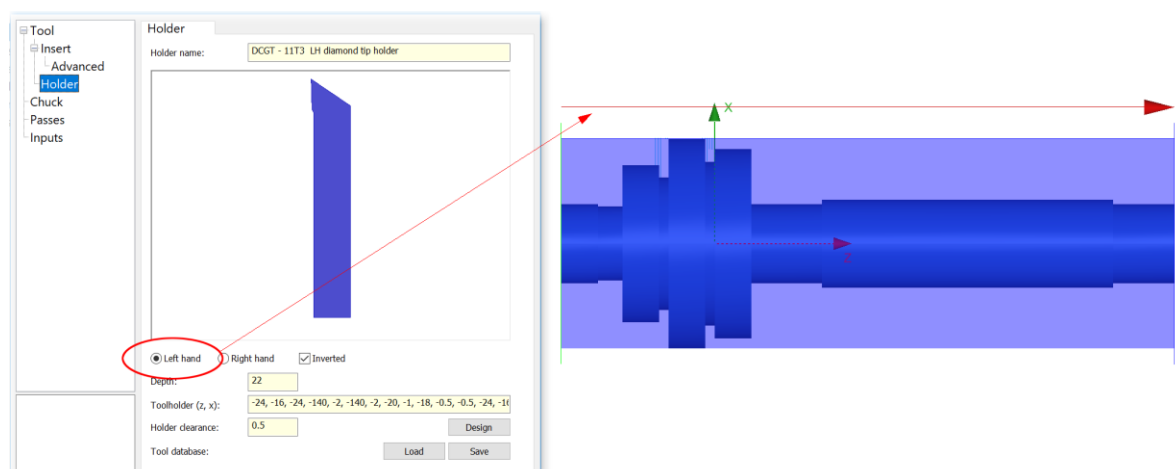
The **Z Min** and **Max** can be set to the groove width. This can be done by rotating the view to a suitable position, clicking in the Z Min or Max, and then clicking of the relevant surface on the part

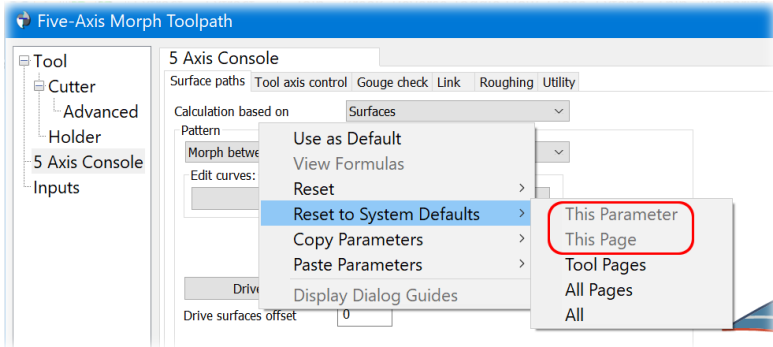
The passes are **linked** to make the toolpath.

**10244: Turning:** The **Rest Roughing** functionality has been added. In keeping with the milling, select the surfaces folder and the previous toolpath folder(s), and Rest roughing will be available from the ribbon.

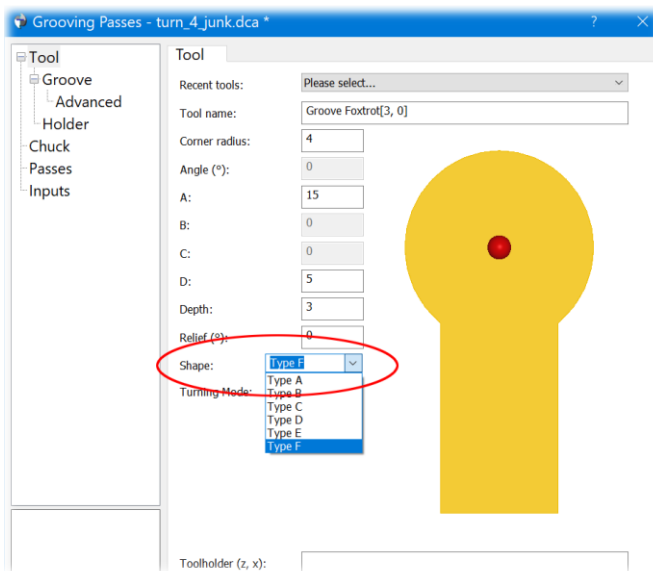


**10249: Turning:** Turning dialog guide graphics have been added to indicate the general machining direction / tool approach direction, depending on Left hand or Right hand tooling having been defined.





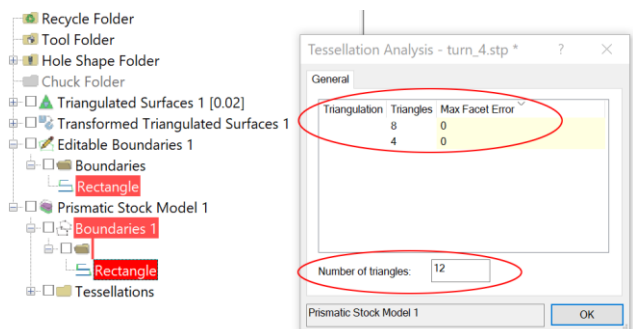
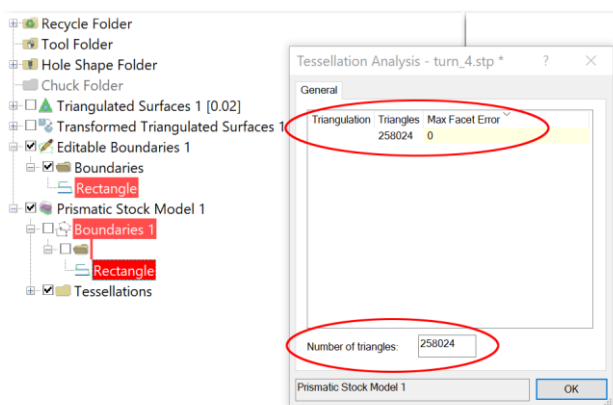
**10258: Turning:** A new **Grooving insert** has been added, a round grooving insert (Type F).



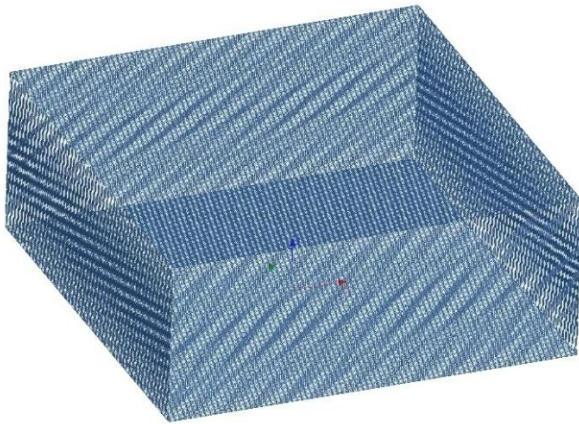
## Stock Model:

**9408: Prismatic Stock Models:** The creation of Prismatic Stock Models has been improved, so to create fewer triangles.

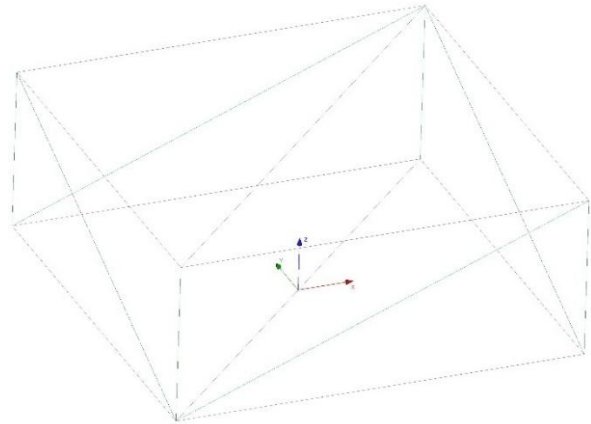
The following images show the number of triangles for a prismatic stock model 100mm high created from a 250mm x 250mm rectangular boundary, in v19.0.15 and v20.0.00



v19.0.15 – 258024 triangles

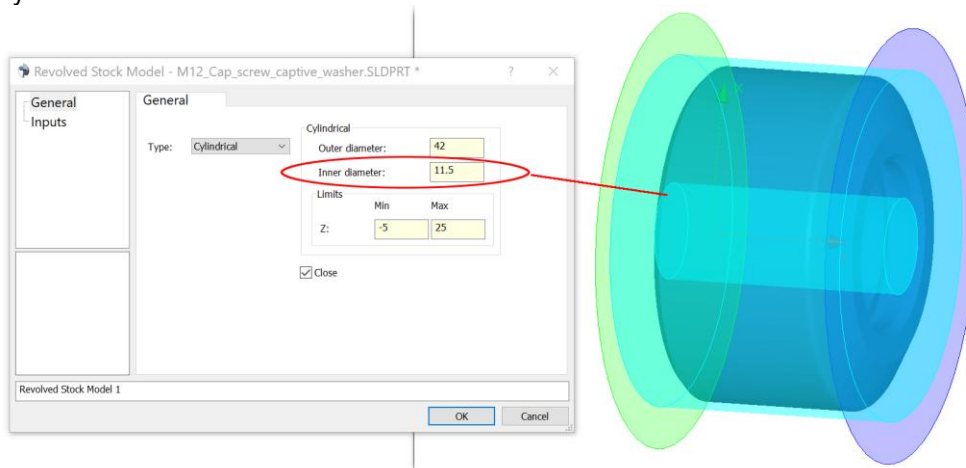


v20.0.00 – 12 triangles



In addition the edges are sharp in v20.

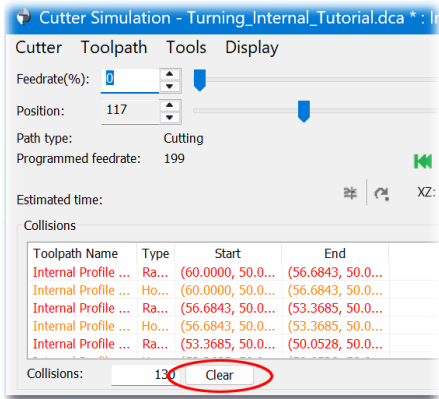
**10188: Stock Model:** A revolved stock model has had an 'Inner diameter' option added, allowing a hollow cylinder to be defined as stock.



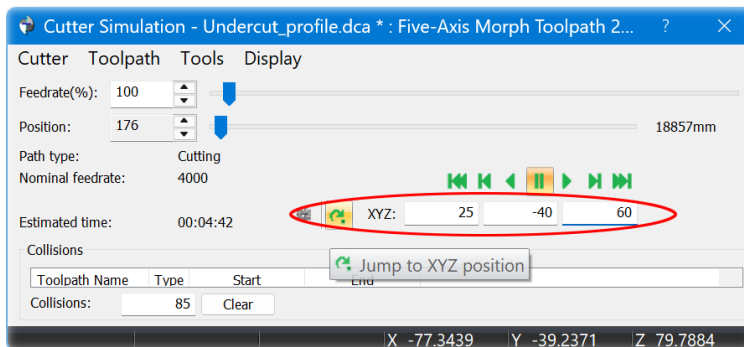
**10171: Transform:** The transformation of 2D Turning Stock Models was not working correctly, this has now been fixed.

## Simulation:

**10471: Cutter Simulation:** A button has been added to the dialog to **Clear** the current list of detected collisions.



**10551: Cutter Simulation:** The Jump to XYZ position has been enabled when using Cutter Simulation, to be in keeping with the Cutter Animation and work in the same way.



**6404: Cutter Simulation:** There were cases that when moving the location slider bar to move the cutter forward would cause parts of the simulation to be missed, this has now been fixed.

## Macros:

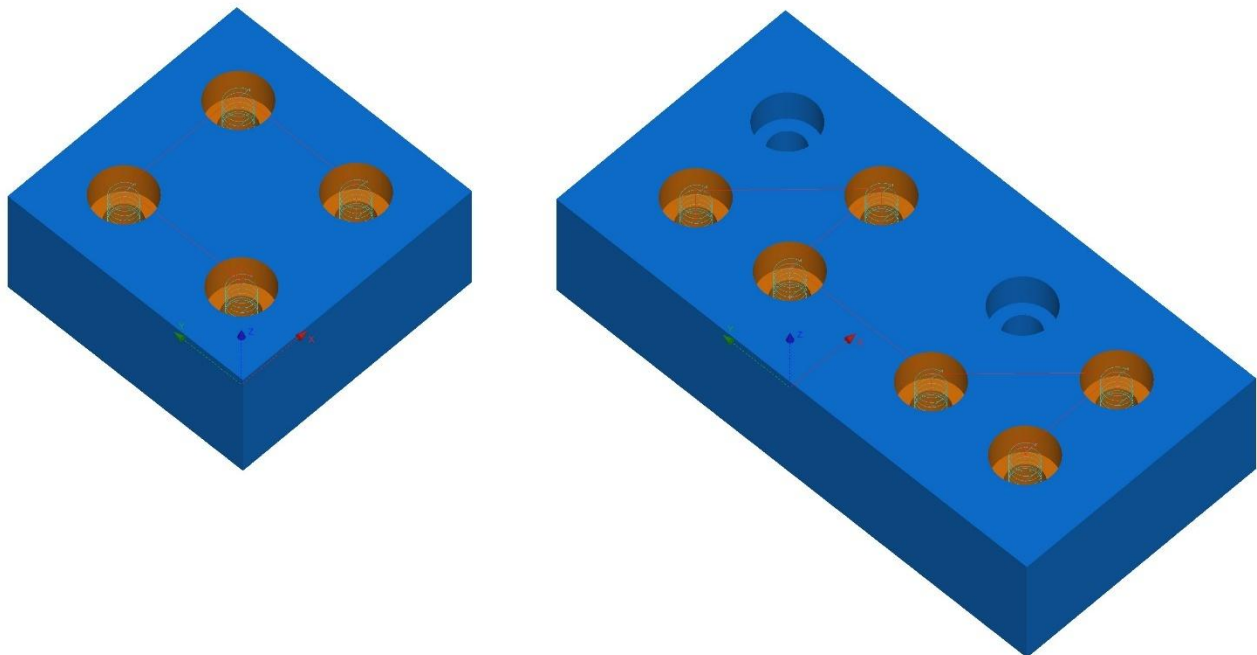
**7111: Macros:** Macros can now be created for complete drilling operations – from hole detection to post-processing. This was not previously possible.

The prerequisite is that the hole shapes are identical, meaning the diameter, depth, and number of segments match. The hole positions may differ. The hole colour is also crucial – only holes with the same shape and colour are considered.

The macro is recorded as usual, without the drilling option being activated.

As before, the geometry file can either be selected when running the macro (File open dialogs only) or loaded into the database before starting the macro (Surface Independent).

In the following example, the same macro was executed on two components. The macro includes hole detection, drilling, and countersinking.



The different hole positions are taken into account. In the component on the right, the blue holes are omitted because they do not have the same color as the holes used to create the macro.

### **Post Processor:**

**10410: Post Processor: ISO, Siemens, Heidenhain and Datron:** For drilling, Reaming, tapping and boring cycles, it was possible for the comments XY thickness 0, Z thickness 0 to be output. However, those cycles have no thickness parameter, so thickness comment is no longer output

Additionally, the Heidenhain post would always output **Thickness XY n.n** and **Thickness Z n.n** on separate lines in the NC file. In keeping with other post processor, it is now possible to use the post option: **Output the XY and Z thicknesses on a single line** to have the thickness comment on one line (it is set false by default so no post will be affected). The **Thickness XY/Thickness Z** was also fixed text in the post, these are now post options: **Text prefix for the XY thickness** and **Text prefix for the Z thickness**, so the user may modify them if wanted.

**10347: Post Processor: Vericut output:** Added 'Y0' in the Holder origin section, as this has been needed since Vericut 9.6.

**10405: Post Processor: Heidenhain:** Added the new option: **BLKFRM size from 1st toolpaths Protected surfaces** (a prismatic stock model representing the billet material). This is set false by default so not to modify any existing post processor. When set false the BLK FRM size is taken from the Triangulated surfaces being machined. If set true, any Protected surfaces used as an input to the first toolpath, the size for the BLK FRM is recovered from the Protected surfaces. If set true, but there are no protected surfaces the Triangulated surfaces are used.

**10309 & 10348: Post Processor: Heidenhain:** Removed two options (Cycle 267 - Surface coordinate prefix and Cycle 267 - Feedrate prefix) related to the external thread milling. They have been replaced by three new parameters: **Depth at front** this is for Q358, **Countersinking offset at front** for Q359, these both get set as 0. And **Countersinking feedrate** for Q254 which is set as the cutting feedrate. It is unlikely that any existing post Processor is affected by this change, but if you do external thread milling, please check the first external thread milling toolpath posted. Updated the Help to match the changes.

**9867: Post processor: Heidenhain:** Added support for spindle speed changes for the Adaptive Clearance toolpaths helix moves.

**10090: Post Processor: Master ISO, Master Siemens, and Master Heidenhain:** Added options for a tool summary list. It is now possible to output a summary for tools used in the master file. The additional tool summary options are:- **Output summary of tools used**, **Text for start of tool summary**, **Title for summary of tools used**, **Tool summary Tool name prefix**, **Display word for tool summary cutter: Diameter**, **Tool summary separator for Diameter and Radius**, **Display word for tool summary cutter: Radius**, **Output Tool summary cutter details with tool name**, **Text for end of tool summary**. These are in the Toolchange section of the post options. By default, the **Output summary of tools used** is set **false** so existing post Processor will not be affected.

Options

| Parameter                                            | Formula         | Type   | Variable            |
|------------------------------------------------------|-----------------|--------|---------------------|
| <b>TOOL SETTINGS</b>                                 |                 |        |                     |
| 60 Output summary of tools used                      | false           | bool   | bOutputSummaryOf    |
| 61 Text for start of tool summary                    | ""              | string | strToolSummaryStar  |
| 62 Title for summary of tools used                   | "Tool summary:" | string | strToolSummaryTitle |
| 63 Tool summary Tool name prefix                     | " T: "          | string | strToolSummaryTool  |
| 64 Display word for tool summary cutter: Diameter    | " Diameter "    | string | strToolSummaryCuti  |
| 65 Tool summary separator for Diameter and Radius    | " x "           | string | strToolSummaryCuti  |
| 66 Display word for tool summary cutter: Radius      | " Radius "      | string | strToolSummaryCuti  |
| 67 Output Tool summary cutter details with tool name | false           | bool   | bOutputToolSumma    |
| 68 Text for end of tool summary                      | ""              | string | strToolSummaryEnd   |
| 69 Output toolchange in Master program               | true            | bool   | bToolChangeInMast   |
| 70 Tool number prefix                                | "T"             | string | strToolNumberPrefix |
| 71 Tool number suffix                                | ""              | string | strToolNumberSuffix |
| 72 Output Tool name not Tool number                  | false           | bool   | bOutputToolName     |
| 73 Toolchange command                                | "M6"            | string | strToolChange       |

**9880: Post Processor: Datron NEXT:** Fixed an issue where if 'Output tool name not tool number' was set false, the tool name was still output, not the tool number.

**10455: Post Processor: ISO Siemens:** If the **Stop coolant between toolpaths** was set **true**, the coolant could be output twice for the first toolpath but omitted for the following toolpaths with the same tool number. This has been fixed.

**10300: Post Processor: G-Post APT:** Fixed issues where if the spindle direction was set counter-clockwise, and the spindle speed was reduced for helical ramping, and unwanted 'spindle stop' was output, and the spindle would then be started in the wrong direction.

**10236: Post Processor:** The MasterFileHeidenhain was not outputting the **Machine safe text before sub program call**, this has been fixed and the default text has been made empty, so no existing post should be affected. There is also a new **Machine safe text after sub program call** option that has been added for MasterFileHeidenhain, MasterFileISO, and MasterFileSiemens. The default text is empty so existing posts will not be affected.

### Import:

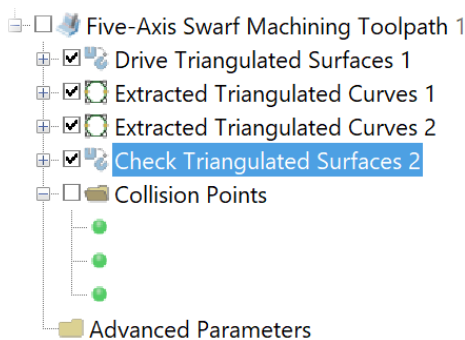
**10469: Import:** The Datakit libraries have been updated to 2025.4.

**9707: Import:** Support for loading and post processing CLD (Toolmaker, Cutter Location Data) files has been removed.

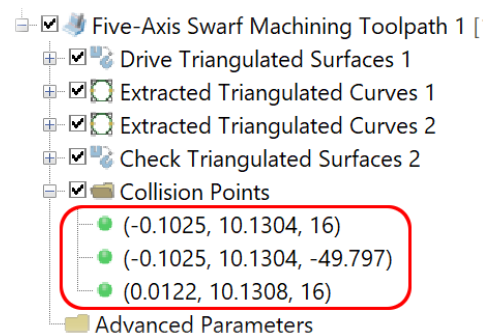
### Five-Axis:

**10033: Five-Axis:** The ModuleWorks libraries have been updated to MW2024-12.

**10204: Five-Axis:** The **collision points** in the tree view now have a label with the co-ordinates of the collision.



The collisions points folder in v19.0,15



The collisions points folder in v20.0.00

**10301: Five-Axis:** The **Automatic** button has been added to the Clearance area options in the NCG CAM dialog.

