

NCG CAM

NCG CAM Solutions Ltd.

NCG CAM - V17.0

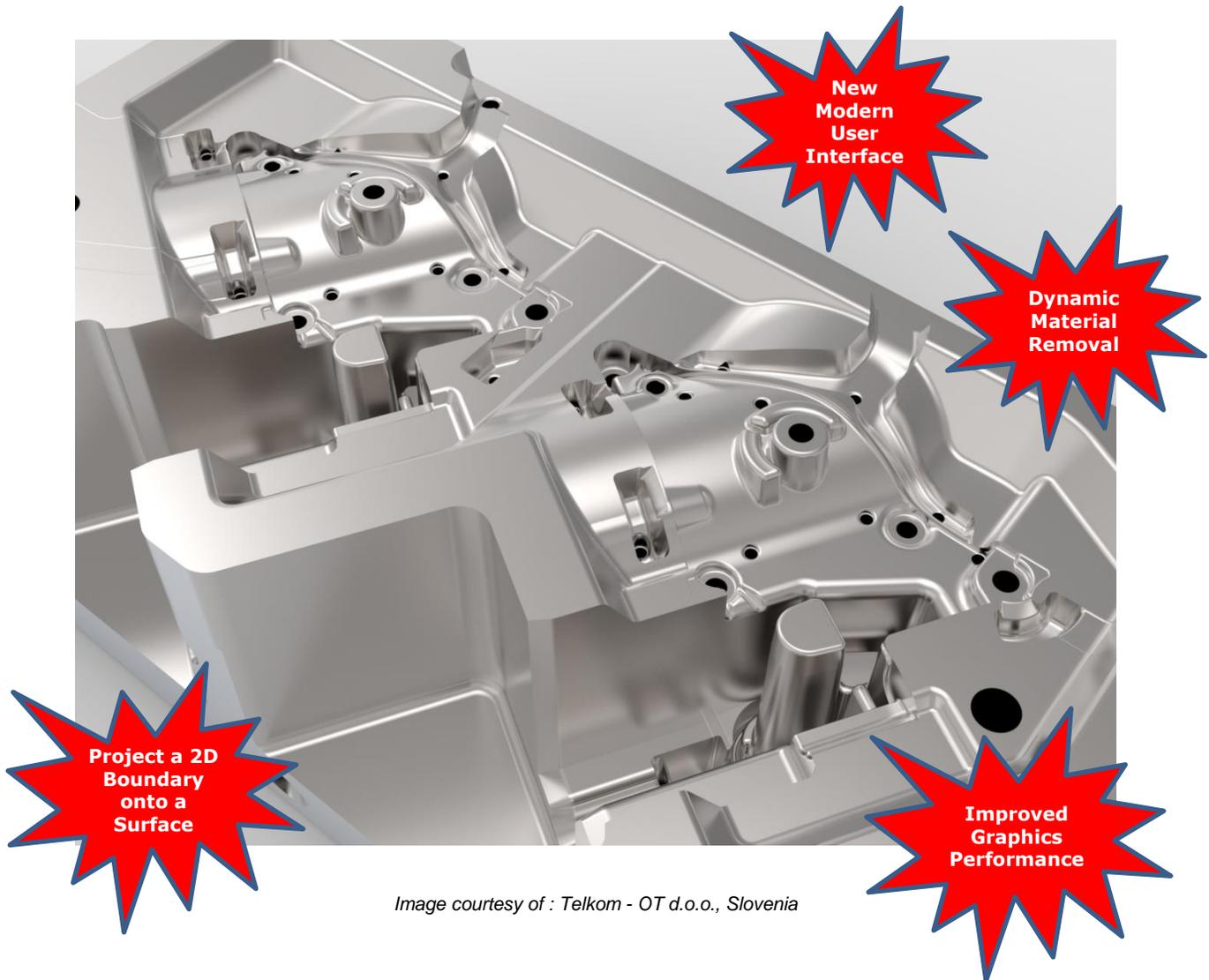


Image courtesy of : Telkom - OT d.o.o., Slovenia

**New Features for NCG CAM
V17.0**

NCG CAM New Features V17.0– Base Module

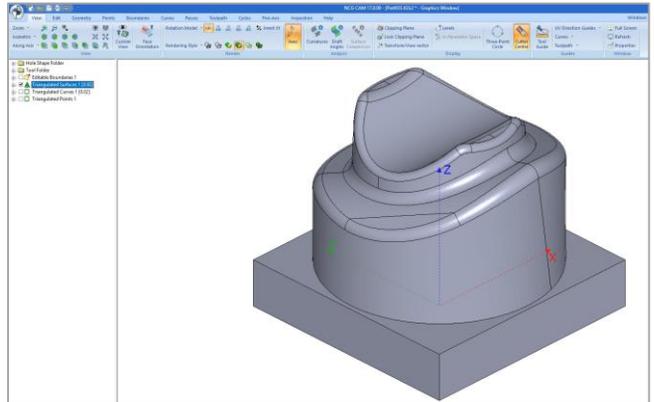
New Modern User Interface

The basic style of the **NCG CAM** User Interface has remained unchanged since this CAM software was first produced 23 years ago.

The **NCG CAM** v17 User Interface now has a Ribbon Design. Ribbons are the modern way to help Users find, understand, and use commands efficiently.

The command bar organises the program's features into a series of tabs at the top of a window; because of the ribbon layout it should be easier for the User to discover new features and functions. All the icons have been redesigned, which adds to the totally fresh new look and feel of the software.

For Existing Users; there is currently a button which will allow them to revert to the Original User Interface. This gives Users the ability to keep up with their existing workload and migrate to the new User Interface as time permits.



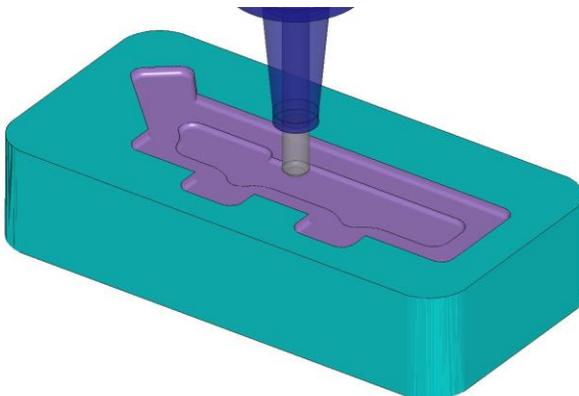
New Modern User Interface:

*Above– View Tab
 Below – Toothpath Tab and Passes Tab*



Dynamic Material Removal

Dynamic Material Removal has been added alongside the existing cutter animation function.

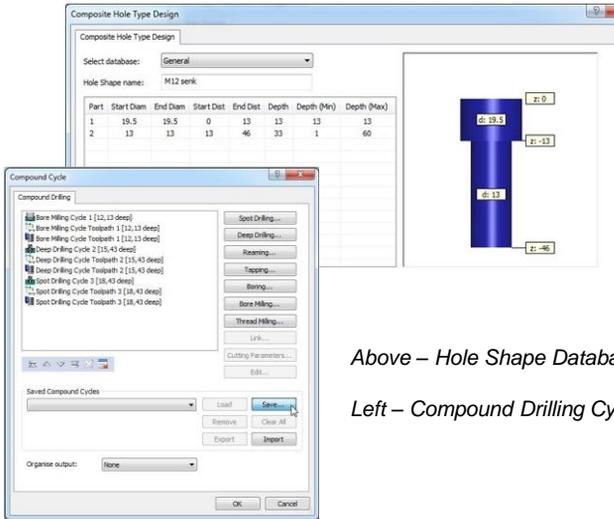


It can indicate the direction of milling, Climb or Conventional, and is able to identify any Gouges or Collisions.

This will enable better visualisation and verification of the material stock removal produced by the calculated toolpath.

Left – Dynamic Material Removal

Drilling - Hole Shape Database and Compound Drilling Cycles



Above – Hole Shape Database
 Left – Compound Drilling Cycles

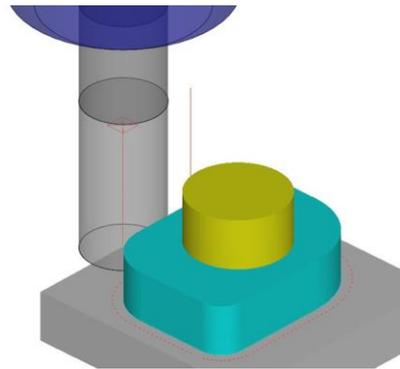
In order to further automate the generation of drilling cycles, it is now possible to create a database of hole shapes and to have the Drilling Data Folders searched for those hole shapes. Furthermore, several cycles can be grouped together as compound cycles, to apply them to specific hole shapes.

The Compound Drilling Cycle can also be applied to holes of different directions (tool axis), so that holes with different directions but the same hole shape, can be programmed in one step.

Along Curve Passes - Displaying Cutter Compensation

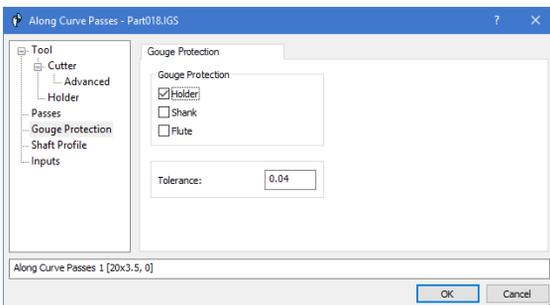
When Cutter Compensation is applied (left or right) to the Along Curve Passes machining routine, after adjusting for the compensation, the proposed cutter path centre line is now displayed as a dotted line.

The left or right compensation is dependent on the direction of the Curve, direction arrows are graphically drawn on the curve while the passes dialog is available, so that the user can amend the direction if required. The drawing of the centre line of the cutter will give visual confirmation of the actual toolpath offset.



Above – Along Curve Passes – Cutter Compensation

Along Curve Passes – Gouge Protection

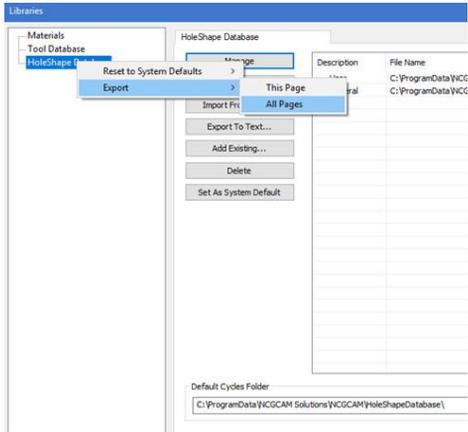


Above – Along Curve Passes – Gouge Protection

The passes dialog for Along Curves now contains three different gouge protection options, the Holder, the Shank and the Flute; each of these can now be set independently.

For example – Machining curves for an engraving routine would effectively require the flute of the cutter to be allowed to gouge, while the holder and shaft could be protected.

Export Option Settings



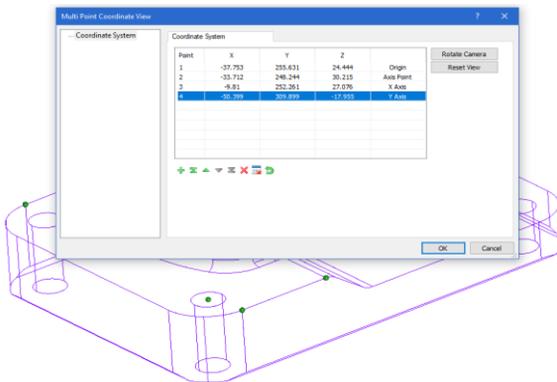
Users can now Export the Tools > Options > Settings. This is via context menus for pages on the Tools Options and Libraries dialogs.

Each menu has "This Page" and "All Pages" options. The export creates a registry editor (.reg) file that will restore the contents of the page.

This will be particularly useful when Users are upgrading to a new major version of **NCG CAM** or installing the software onto a new computer.

Left – Export Option Settings

Local Coordinate System



The Coordinate System entity has been extended to include the Axis rotation information. The User can create a points folder containing key reference points and defining the orientation. By selecting this Coordinate System folder and a Triangulated Surfaces folder or toolpath in the Directory Tree, then selecting the option 'Set Coordinate System' it will reposition the part/s or toolpath/s to the selected plan.

Where multiple parts are imported, this will reduce the time to relocate them to a common Coordinate System .

Left – Local Coordinate System

Project a 2D Boundary onto the Surface

3D curves can now be projected from a 2D boundary. This is meant as a visualisation aid to help with the creation of additional containment boundaries.

This is mainly aimed at 3+2 axis machining as existing boundaries are drawn above the surfaces along the tool axis.

Prior to this it was difficult to be sure of the exact position of contact of a boundary on the surfaces if you are working from a different tool axis



Right – Project a 2D Boundary onto the Surface

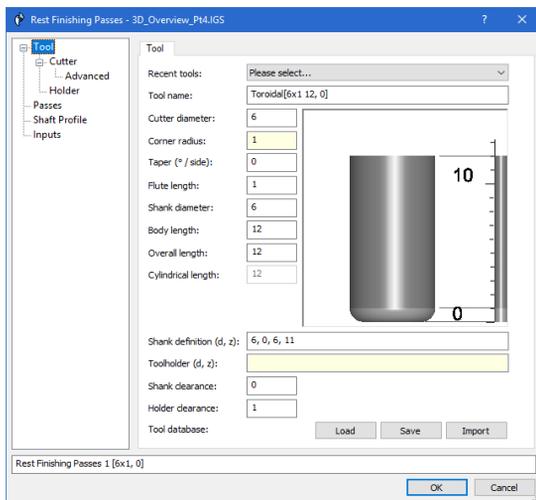
Improved Graphics Performance

The internal graphics library has been over-hauled to give a significant performance boost to the speed of dynamic rotation and translation of shaded models that contain large numbers of triangles; **NCG CAM v17** is now 30 – 40% faster than **NCG CAM v16**.

The large number may be due to the large physical size of the model or a tight triangulation tolerance required for machining.

Previously the user may have needed to set the 'Dynamic Rendering Style' to one of the 'Wireframe' options, to better handle the rotation of these bigger models, this of course may still offer benefits.

Rest Finishing – Support Toroidal Target Cutters



This new feature gives the ability to use Toroidal cutters to machine the Rest Finishing areas.

Previously Ball-nosed and Toroidal cutters could be used as Reference cutters to find the corners of the job, but the Target cutter had been restricted to a Ball-nose.

****This feature will not be available in NCG CAM v17.0 but will follow on in a v17 point release once completed.**

Left – Rest Finishing – Support Toroidal Target Cutters

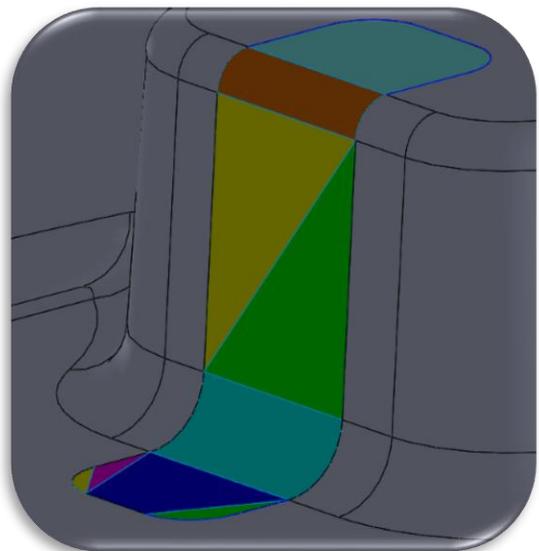
Surfaces - Create a Patch from Two 3D Curves

This development will provide additional CAD functionality, allowing the User to Create a Surface Patch from two 3D curves.

If the curves have coincident start points, then the surface created would resemble a triangular patch.

If the two curves are spaced apart, this would create a rectangular patch.

****This feature will not be available in NCG CAM v17.0, but will follow on in a v17 point release once completed.**



Right – Create a Patch from Two 3D Curves

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