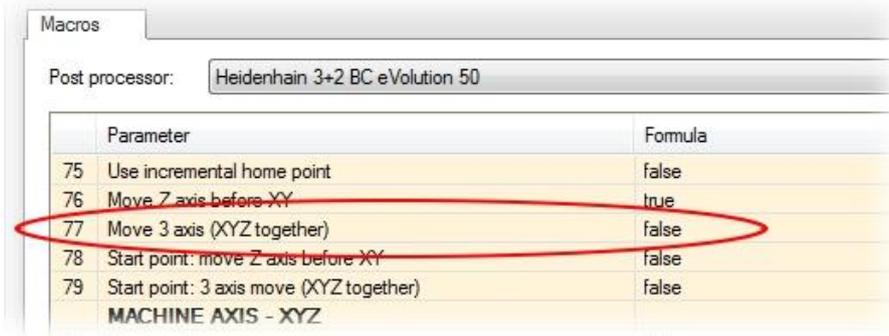


NCG CAM Solutions Ltd are pleased to release NCG CAM v14.0.03

V14.0.02 was release just over 2 months ago, with this point release there are some little fixes, and some enhancements to existing features.

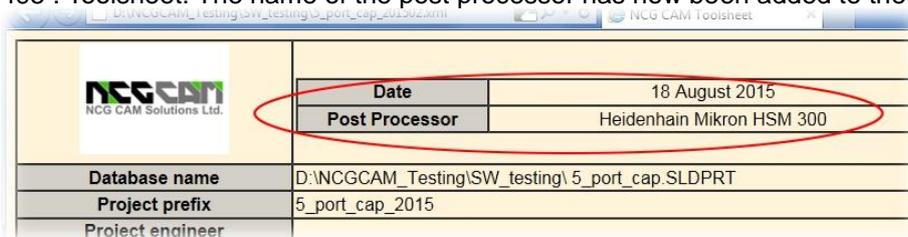
Please note that NCG CAM v14.0 will not install on Window XP, and that at the end of this year NCG CAM will be 64 bit only.

463 : Post Processor: Heidenhain: The option of moving the tool in the X, Y and Z simultaneously to go to the home position for the Heidenhain Post Processor has been added. The option 'Move 3 axis (XYZ together)' is set 'false' by default so existing posts are not affected.



Parameter	Formula
75 Use incremental home point	false
76 Move Z axis before XY	true
77 Move 3 axis (XYZ together)	false
78 Start point: move Z axis before XY	false
79 Start point: 3 axis move (XYZ together)	false
MACHINE AXIS - XYZ	

495 : Toolsheet: The name of the post processor has now been added to the toolsheet.



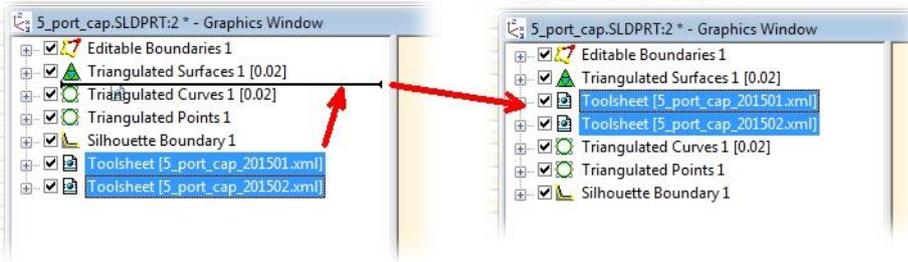
	Date	18 August 2015
	Post Processor	Heidenhain Mikron HSM 300
Database name	D:\NCGCAM_Testing\SW_testing\5_port_cap.SLDPRT	
Project prefix	5_port_cap_2015	
Project engineer		

1321 : Analysis functions: The initial response time for surface comparison analysis has been improved.

3422 : Post Processor: There was a problem when posting a transformed toolpath when it had been transformed using one of the pre-defined macros (cx(), mx(), my() etc) or using the 27 point grid, was giving an error dialogue, this has now been fixed.

3649 : Thread Milling: The default 'Set Up Clearance' has been changed from 1mm to 1mm + Pitch/2, this allows extra clearance for when the tool length has to be set to the centre of the tip.

3851 : User Interface: Multiple selected plans can now be moved by drag and drop within the tree view.



Select the items to be moved within the contents tree (they do not need to be consecutive), then with Shift+Mouse left drag to the new position within the contents tree

3859 : Machining Passes: Point spacing default value is now zero when Max segment length is not enabled.

3892 : Curves: Fixed problem when using autoplanarise on rotated curves. The curves are not on the correct plane.

3895 : Helical Machining: Correct handling of negative thicknesses

3900 : User Interface: Make constraint boundaries an optional input to Five Axis Along Curve, Swarf, Parallel Cuts and Morph plans.

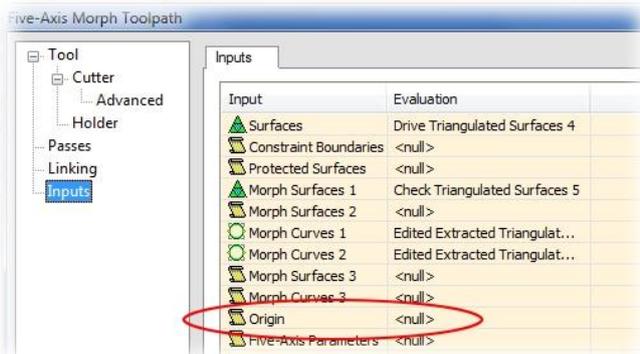
3907 : User Interface: There was a problem when using the tool guide with multiple rotated drilling data folders not setting the guide orientation correctly, this has now been fixed.

3909 : Linking: Horizontal Area Passes: Fixed a case where the Ramp Extension was not being applied when the Lead in radius was zero.

3914 : User Interface: Increased the width of the cutter animation dialog in translated versions to include all buttons.

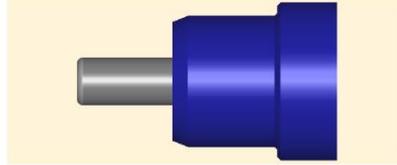
3917 : Post Processing: When post processing with GPOST the extension was not being added to the end of the output filename. This has now been fixed.

3923 : User Interface: It is now possible to add or remove the origin point input from a Five-Axis plan.



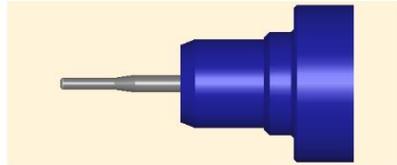
3929 : Toolsheet: A new section has been added to the toolsheet giving an image and details of each tool used. Also fixes #1483

Tool number	1
Tool type	Toroidal
Cutter name	Toroidal[16x2 32, 0.5]
Cutter diameter	16
Corner radius	2
Taper angle	0
Number of teeth	2
Flute length	2
Body length	32
Cylindrical length	32
Tool cutting time	00:18:37
Tool name	Toroidal[16x2 32, 0.5] + BT30 Multi Lock - C16 55 long
Tool holder	BT30 Multi Lock - C16 55 long
Holder details	0 0 40 0 44 5 44 34 46 35 51 35 53 36 53 55



[Back to summary](#)

Tool number	5
Tool type	Toroidal
Cutter name	Toroidal[4x1 40, 0.5]
Cutter diameter	4
Corner radius	1
Taper angle	0
Number of teeth	2
Flute length	1
Body length	40
Cylindrical length	18
Tool cutting time	00:06:13
Tool name	Toroidal[4x1 40, 0.5] + BT30 Multi Lock - C12 55 long
Tool holder	BT30 Multi Lock - C12 55 long
Holder details	0 0 26 0 30 5 30 28 31 28 35 30 35 37 37 38 51 38 53 39 53 58



This addition should be useful for preparing tools and holders prior to running the machine. If you do not want the tool summary, it is possible to disable it, then there is a switch in the Extended toolsheet producer or Classic plus toolsheet producer.

Parameter	Formula	Type	Variable
1	Default filename extension	string	strFileExtension
2	Number of decimal places	int	nDecimalPlaces
3	Decimal separator	string	strDecimalSeparator
4	Style sheet	string	strXMLStyleSheet
5	Company logo	string	strCompanyLogo
6	NCG CAM logo	string	strApplicationLogo
7	Small company logo	string	strCompanyLogoSmall
8	Small NCG logo	string	strNCGLogoSmall
9	Use relative pathnames	bool	bRelativePathnames
10	Include tool summary	bool	bIncludeToolSummary

3931 : Toolsheet: The toolsheet was giving the wrong tool type for drilling & advanced Five Axis cutters this has now been fixed.

3934 : Security: A check has been added to determine if the dongle in use is of a suitable configuration to be used, or whether it should be swapped out for a new one. Depending on the internals of your dongle, a message may be displayed, if displayed it will ask you to contact your reseller and let them know the dongle number. Then a swap out dongle can be arranged before the end of the year.

3935 : Security: Updated the Sentinel-Protection-Installer to version 7.6.8.

3937 : Correctly report operating system version on Windows 10.

3939 : Analysis functions: The initial response time for the analysis of draft angles has been improved.

3941 : Corrected the ModuleWorks dialog and simulator German DLLs.

3953 : Thread Milling: Internal threads: As default the thread mill should be of smaller diameter than the hole. Added new validation warning.

3965 : User Interface: When using the Control key with Convert To Curves option the Advanced dialog did not get displayed, this has now been fixed.

Documentation: Updated the documentation.

Updated the German, Japanese TMJ and Turkish translations.