PRODUCT RELEASE ANNOUNCEMENT

Product Category	**CADverter**
Product Group	NX for CATIA V5 Multi-CAD
Release Versions	19.3

Document Type	**Product Release Announcement**
Status	Released
Revision	3.0
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Issued	22/09/2016

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History

Revision	Update Information
1.0	Initial Release
3.0	Added NX releases

Product Codes

Product Code	Product Type	Product Description
MC5NX-B1	Base Product	CATIA V5 (Multi-CAD) to/from NX (bi-directional)
MC5NX-U1	Base Product	CATIA V5 (Multi-CAD) to NX (uni-directional)
MC5NX-A1	Optional add-on	CATIA V5 (Multi-CAD) to NX (Bi-directional) Model Based Definition (3D Dimensions & Annotations)
MC5NX-A2	Optional add-on	CATIA V5 (Multi-CAD) to NX (uni-directional) Model Based Definition (3D Dimensions & Annotations)
NXMC5-U1	Base Product	NX to CATIA V5 (Multi-CAD) (uni-directional) Model Based Definition (3D Dimensions & Annotations)
NXMC5-A1	Optional add-on	NX (Bi-directional) to CATIA V5 (Multi-CAD) Model Based Definition (3D Dimensions & Annotations)
NXMC5-I1	Optional add-on	NX to CATIA V5 (Multi-CAD) (uni-directional) Interactive.

Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V5	CATIA V5-6R2013 (SP4 minimum recommended) CATIA V5-6R2014 (SP4 minimum recommended) CATIA V5-6R2015 CATIA V5-6R2016
NX	8.5 9.0 10.0

Supporting API's

The following supporting API support is used

Application	Revision
Dassault Multi-CAD (XCAD)	CATIA V5-6R2013 CATIA V5-6R2014 CATIA V5-6R2015 CATIA V5-6R2016

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Windows 7 64bit	Qualified and fully supported
Window 8 64bit	Qualified and fully supported for CATIA V5-6R2013, CATIA V5-6R2014 CATIA V5-6R2015 and CATIA V5-6R2016
Windows Server 2008 R2	Qualified and fully supported for CATIA V5-6R2013, CATIA V5-6R2014 CATIA V5-6R2015 and CATIA V5-6R2016
Windows Server 2012	Not formally qualified for CATIA V5-6R2013 Qualified and fully supported for CATIA V5-6R2014 CATIA V5-6R2015 and CATIA V5-6R2016 only

Distribution Media

CD images of the latest release are available from the following download sites.

Note! For each CATIA V5 release the CATIA V5 Multi-CAD software requires the relevant Dassault Systems TXX media to be downloaded and installed prior to installing the Theorem Multi-CAD Product Software. Please see the Software Installation guide for further details.

Product	Select hyperlink to download installation CD
TXX for CATIA V5-6R2013	Download TXX Platform Media for CATIA V5-6R2013
TXX for CATIA V5-6R2014	Download TXX Platform Media for CATIA V5-6R2014
TXX for CATIA V5-6R2015	Download TXX Platform Media for CATIA V5-6R2015
TXX for CATIA V5-6R2016	Download TXX Platform Media for CATIA V5-6R2016
Multi-CAD CATIA V5-6 – NX for all supported CATIA V5 releases	Download CAD_19.3_MC5NX_WIN.01 Release
Generic Theorem User Interface	Download CAD_19.3_UI_WIN.02 Release

New Features / Enhancements

The following new features or enhancements have been introduced with this release

Product Impacted	New Feature / Enhancement Description
NX to CATIA V5-6	Improved PMI Positioning
NX to CATIA V5-6	Support for the translation of reference sets
NX to CATIA V5-6	Improved read of Product structure
NX to CATIA V5-6	Improved geometry fixup capability
NX to CATIA V5-6	Improved reading / creation of tessellated bodies
NX to CATIA V5-6	Improved support for mass property reading
NX to CATIA V5-6	Improved performance of PMI processing
NX to CATIA V5-6	Improved processing of invalid solids
NX to CATIA V5-6	Fix XCAD server memory allocation problem
NX to CATIA V5-6	Added new license check for PMI processing.
NX to CATIA V5-6	Change to the code that determines the "up" vector for a view to line up with the latest Siemens documentation.
CATIA V5-6 to JT	Correction to the creation of Coordinate Systems.
CATIA V5-6 to JT	Allow user control of exporting V5 hidden geometry
CATIA V5-6 to JT	Fix for 3D annotations read from some 3DXML files.
Multi-CAD Architecture	Implemented new read/write architecture
Multi-CAD Batch	Added separate _BATCH file processing license to control batch processing usage
Multi-CAD Batch	Add simplified command line options -o <outfile> mode_visu mode_visu_snap mode_catpart
Multi-CAD Menu	New optional CATIA Menu item File > Theorem Command. Plus environment variable to control the process that is launched
Multi-CAD Menu	Enhance generation of progress files to include suitable XML format output that is used by the Theorem User Interface.

Known Limitations and Restrictions

The following limitations and restrictions have been identified during final testing prior to release

Ref ID	Limitations and Restrictions
1	*Minimum Service Pack + Hotfix Level:* There are a number of issues related to the use of the Multi-CAD infrastructure with lower Service Pack levels of the CATIA V5 installation. See above for require minimums. For the latest Hotfix information please contact your local DS support agent.
2	*Running the NX to V5 translator via the Theorem UI:* There is a known limitation imposed with this release which means that both the creation of VISU & VISUSNAP output format data is not supported when using the Theorem UI. This restriction is NOT imposed when either running the translator interactively within V5 or when the process is run from the command line. Therefore please limit the use of the Theorem UI for NX to V5 when creating geometry in CATPart format.
3	*Some limitations when processing Model Views with PMI from NX to V5*

Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID#	Product Impacted	Description