



PRODUCT RELEASE ANNOUNCEMENT

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Product Group	CREO for CATIA V5 Multi-CAD
Release Versions	19.3

Document Type	Product Release Announcement
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Contents

History	2
Product Codes	3
Supported Application Revisions.....	4
Supporting API's	4
Supported Operating Systems.....	4
Distribution Media.....	6
New Features / Enhancements	7
Known Limitations and Restrictions.....	8
Resolved Support Calls	9



History

Revision**Update Information**

1.0	Initial Release

Product Codes

Product Code	Product Type	Product Description
MC5CRE-B1	Base Product	CATIA V5 (Multi-CAD) to/from NX (bi-directional)
MC5CRE-A1	Optional add-on	CATIA V5 (Multi-CAD) to CREO (Bi-directional) Model Based Definition (3D Dimensions & Annotations)
MC5CRE-U1	Base Product	CATIA V5 (Multi-CAD) to CREO (uni-directional)
MC5CRE-A2	Optional add-on	CATIA V5 (Multi-CAD) to CREO (uni-directional) Model Based Definition (3D Dimensions & Annotations)
CREMC5-U1	Base Product	CREO to CATIA V5 (Multi-CAD) (uni-directional) Model Based Definition (3D Dimensions & Annotations)
CREMC5-A1	Optional add-on	CREO (Bi-directional) to CATIA V5 (Multi-CAD) Model Based Definition (3D Dimensions & Annotations)



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
CREO	CREO Parametric 2.0 CREO Parametric 3.0

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	CATIA and Creo - Qualified and fully supported all versions
Window 8 64bit	CATIA: <ul style="list-style-type: none"> • Not formally qualified for CATIA V5 R19, R20, R21, V5-6R2012 • Qualified and fully supported for V5-6R2013, V5-6R2014, V5-6R2015, V5-6R2016 Creo: 2.0, 3.0
Windows Server 2008 R2	CATIA: <ul style="list-style-type: none"> • Not formally qualified for V5 R19, R20 • Qualified and fully supported for R21, V5-6R2012, V5-6R2013, V5-6R2014, V5-6R2015, V5-6R2016 Creo: <ul style="list-style-type: none"> • Qualified and fully supported for 2.0 • Not formally qualified for 3.0



Windows Server 2012	CATIA: <ul style="list-style-type: none">• Not formally qualified for V5 R19, R20, R21, V5-6R2012, V5-6R2013• Qualified and fully supported for V5-6R2014, V5-6R2015, V5-6R2016 Creo: <ul style="list-style-type: none">• Qualified and fully supported for 2.0 M150 or later and 3.0



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 – CREO for all supported CATIA V5 releases	Download CAD 19.3 MC5CRE 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
CREO to CATIA V5-6	
CREO to CATIA V5-6	
CATIA V5-6 to CREO	
CATIA V5-6 to CREO	
CATIA V5-6 to CREO	
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	<i>Minimum Service Pack + Hotfix Level:</i> 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	<i>Running the CREO to V5 translator via the Theorem UI:</i> 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	<i>Some limitations when processing Model Views with PMI from CREO to V5</i>
4	<i>CREO to V5 translator via CATIA “Insert Existing”:</i> Visu and VisuSnap only process assemblies. If translating a part file, an empty CGR file is created.



Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID#	Product Impacted	Description
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