

PRODUCT RELEASE ANNOUNCEMENT

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History

Revision	Update Information
1.0	Initial Release
2.0	Appending Software links and Features





Product Codes

CA5NX-B1	CATIA V5 (CAA Based) to/from NX (bi-directional)
CA5NX-U1	CATIA V5 (CAA Based) to NX (uni-directional)
CA5NX-A1	CATIA V5 (CAA Based) to > NX (uni-directional) with 3D PDF Export Add On
CA5NX-A2	CATIA V5 (CAA Based) to > NX (uni-directional) with PMI Add On
CA5NX-A3	CATIA V5 (CAA Based) to <> NX (bi-directional) with PMI Add On
NXCA5-U1	NX to CATIA V5 (CAA Based) (uni-directional)
NXCA5-A1	NX to CATIA V5 (CAA Based) (uni-directional) with PMI Add On
NXCA5-A2	NX to CATIA V5 (CAA Based) (uni-directional) with 3D PDF Export Add On





Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V5	CATIA V5 R19, R20, R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014, CATIA V5-6R2015 & , CATIA V5-6R2016
NX	NX 8.5, NX 9.0, NX10
Adobe Reader	Adobe Reader XI

Supporting API's

The following supporting API support is used

Application	Revision
3D PDF API	HOOPS Exchange Publish 9.1

Supported Operating Systems

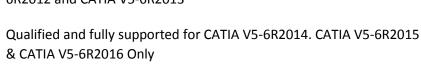
Operating System Qualification Status

The following operating systems have been qualified with this release

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Windows 7 64bit	Qualified and fully supported (All CATIA V5 Revisions)
Window 8 64bit	Not formally qualified for CATIA V5 R19, R21 and CATIA V5-6R2012

Qualified and fully supported for CATIA V5-6R2013, CATIA V5-6R2014
CATIA V5-6R2015 & CATIA V5-6R2016 Only

	CATIA V5-6R2015 & CATIA V5-6R2016 Only
Windows Server 2008 R2	Not formally qualified for CATIA V5 R19 Qualified and fully supported for CATIA V5 R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014, CATIA V5-6R2015 & CATIA V5-6R2016 Only
Windows Server	Not formally qualified for CATIA V5 R19, CATIA V5 R21, CATIA V5-6R2012 and CATIA V5-6R2013





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Distribution Media

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

Product	URL for CD image download	
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Support for ALL Versions	cdCAD 19.3 CA5NX WIN.01.zip
Unified Interface	cdCAD_19.3_UI_WIN.01.zip





New Features / Enhancements

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

Ref New Feature / Enhancement Description

ID

1	V5 INSTANCE RENAMING – Support for the control of how instance are renamed, using a set configuration or the users defined options in the CATSettings
2	V5 REPRESENTATIONS SUPPORT - Allows all representations for a node to be read
3	V5 LEAF NODE PROCESSING - Reads extra level of specification tree structure
4	V5 SELECTIVE PMI PROCESSING – Allows the user to selectively process PMI (read just part PMI, Read part pmi that's in an Assembly set, Reads assembly & part pmi that's in an Assembly set)
5	V5 GEOMETRY EDGE PROCESSING - Allows the edge curves for a planar FTA construction geometry to be created as standalone curves allowing the plane to be viewed edge on
6	V5 ENHANCED COLOUR PROCESSING – Additional support for model colour processing and support for Washes colour specified on the root node of the specification tree down over all bodies
7	V5 WIREFRAME THREAD CREATION - creates the wireframe geometry associated to a FTA Thread construction geometry
8	V5 CONTROL FOR QUASI CYCLICITY - Allows control of the tolerance that a surface is checked for quasi cyclicity
9	V5 DATUM FACE SIZE CONTROL – This allows planar datum geometry to be read as a face and sized to a user defined size.
10	MERGING OF CGR VERTICES – New option that enables the merging of CGR vertices, this is now off by default
11	V5 PMI WRAPPER STATUS - Now installed by default
12	V5 PMI READ UPDATES - Now notes positional and orientation links, PMI – PMI association for GDT's, Simple Datum's & Datum Target's
13	V5 ATTRIBUTE UPDATE - Added units to attributes, support for getting attribute magnitude from attribute mapping file





14	V5 UPDATED COLOUR AND OPACTIY SUPPORT - Part root colour and opacity is now stored on detail. Also now allow opacity on solids / faces to be inheritted from root node of specification tree
15	V5 OUTPUT MBD UPDATES - Axis systems added to groups when this option is on. Option now listed in the option list when used.
16	V5 FTA SCALING UPDATE - Update radius if FTA scaled and start and end angle if rotated
17	V5 MULTILINE NOA PROCESSING - Improvements in reading multiline text in NOA in regard to positioning and justification
18	V5 UPDATE TO CGR PROCESSING - When reading a CATProduct that references a CGR that's not on the system as from delmia save the CGR file to a temp location, read then delete
19	V5 DATA TARGET PROCESSING – Update to remove any duplicated geometry assoication when also associated to Simple Datum
20	NX PMI READ — Updated code for better processing of PMI
21	V5 WRITE: FAILED FACE PROCESSING – added a number of options to allow a user to configure what is done if a BREP translation fails.
22	V5 WRITING ONE CATPART PER BODY – Allows the user to createa separate CATPart per body





Known Limitations and Restrictions

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

Ref ID Limitations and Restrictions

1	Exporting Viewed Data: When exporting viewed V5 assembly data to NX from the Unified Interface the user must give the output NX file a unique name, in order for the assembly to process correctly.
2	NX Interactive: The NX Interactive menus are omitted from this release
3	Incomplete Message: When processing NX to CATIA V5 data some files report an 'incomplete' error message, when the output is geometrically correct.
4	Capture Views in NX: We do not create a semantic, editable PMI output in V5 from NX, just a graphical representation. Therefore we cannot support processing of capture views from NX to V5





Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID	Brief Description
CAS-02370- N1T9C0	Additional axis systems appear in output
CAS-02499- H3B9B6	Return code returning 0 despite that the data was "incomplete"

