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## PRODUCT RELEASE ANNOUNCEMENT

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Product Category	<b>CADverter</b>
Product Group	CATIA V5 for NX
Release Version	19.3

Document Type	<b>Product Release Announcement</b>
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## History

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

Product Codes

<b>Product Code</b>	<b>Product Description</b>
<b>CA5NX-B1</b>	CATIA V5 (CAA Based) to/from NX (bi-directional)
<b>CA5NX-U1</b>	CATIA V5 (CAA Based ) to NX (uni-directional)
<b>CA5NX-A1</b>	CATIA V5 (CAA Based ) to > NX (uni-directional) with 3D PDF Export Add On
<b>CA5NX-A2</b>	CATIA V5 (CAA Based ) to > NX (uni-directional) with PMI Add On
<b>CA5NX-A3</b>	CATIA V5 (CAA Based ) to <> NX (bi-directional) with PMI Add On
<b>NXCA5-U1</b>	NX to CATIA V5 (CAA Based ) (uni-directional)
<b>NXCA5-A1</b>	NX to CATIA V5 (CAA Based ) (uni-directional) with PMI Add On
<b>NXCA5-A2</b>	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
<b>CATIA V5</b>	CATIA V5 R19, R20, R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014, CATIA V5-6R2015 & , CATIA V5-6R2016
<b>NX</b>	NX 8.5, NX 9.0, NX10
<b>Adobe Reader</b>	Adobe Reader XI

## Supporting API's

The following supporting API support is used

Application	Revision
<b>3D PDF API</b>	HOOPS Exchange Publish 9.1

## Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
<b>Windows 7 64bit</b>	Qualified and fully supported (All CATIA V5 Revisions)
<b>Window 8 64bit</b>	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
<b>Windows Server 2008 R2</b>	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
<b>Windows Server 2012</b>	Not formally qualified for CATIA V5 R19, CATIA V5 R21, CATIA V5-6R2012 and CATIA V5-6R2013  Qualified and fully supported for CATIA V5-6R2014. CATIA V5-6R2015 & CATIA V5-6R2016 Only



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## Distribution Media

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

<b>Product</b>	<b>URL for CD image download</b>
<b>Support for ALL Versions</b>	<a href="#">cdCAD 19.3 CA5NX WIN.01.zip</a>
<b>Unified Interface</b>	<a href="#">cdCAD 19.3 UI WIN.01.zip</a>



## New Features / Enhancements

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

Ref ID	New Feature / Enhancement Description
1	<b>V5 INSTANCE RENAMING</b> – Support for the control of how instance are renamed, using a set configuration or the users defined options in the CATSettings
2	<b>V5 REPRESENTATIONS SUPPORT</b> - Allows all representations for a node to be read
3	<b>V5 LEAF NODE PROCESSING</b> - Reads extra level of specification tree structure
4	<b>V5 SELECTIVE PMI PROCESSING</b> – 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	<b>V5 GEOMETRY EDGE PROCESSING</b> - 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	<b>V5 ENHANCED COLOUR PROCESSING</b> – 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	<b>V5 WIREFRAME THREAD CREATION</b> - creates the wireframe geometry associated to a FTA Thread construction geometry
8	<b>V5 CONTROL FOR QUASI CYCLICITY</b> - Allows control of the tolerance that a surface is checked for quasi cyclicity
9	<b>V5 DATUM FACE SIZE CONTROL</b> – This allows planar datum geometry to be read as a face and sized to a user defined size.
10	<b>MERGING OF CGR VERTICES</b> – New option that enables the merging of CGR vertices, this is now off by default
11	<b>V5 PMI WRAPPER STATUS</b> - Now installed by default
12	<b>V5 PMI READ UPDATES</b> - Now notes positional and orientation links, PMI – PMI association for GDT’s, Simple Datum’s & Datum Target’s
13	<b>V5 ATTRIBUTE UPDATE</b> - Added units to attributes, support for getting attribute magnitude from attribute mapping file



14	<b>V5 UPDATED COLOUR AND OPACTIY SUPPORT</b> - 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	<b>V5 OUTPUT MBD UPDATES</b> - Axis systems added to groups when this option is on. Option now listed in the option list when used.
16	<b>V5 FTA SCALING UPDATE</b> - Update radius if FTA scaled and start and end angle if rotated
17	<b>V5 MULTILINE NOA PROCESSING</b> - Improvements in reading multiline text in NOA in regard to positioning and justification
18	<b>V5 UPDATE TO CGR PROCESSING</b> - 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	<b>V5 DATA TARGET PROCESSING</b> – Update to remove any duplicated geometry assoication when also associated to Simple Datum
20	<b>NX PMI READ</b> – Updated code for better processing of PMI
21	<b>V5 WRITE: FAILED FACE PROCESSING</b> – added a number of options to allow a user to configure what is done if a BREP translation fails.
22	<b>V5 WRITING ONE CATPART PER BODY</b> – 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	<i>Exporting Viewed Data:</i> 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	<i>NX Interactive:</i> The NX Interactive menus are omitted from this release
3	<i>Incomplete Message:</i> When processing NX to CATIA V5 data some files report an 'incomplete' error message, when the output is geometrically correct.
4	<i>Capture Views in NX:</i> 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

<b>Support Call ID</b>	<b>Brief Description</b>
<b>CAS-02370-N1T9C0</b>	Additional axis systems appear in output
<b>CAS-02499-H3B9B6</b>	Return code returning 0 despite that the data was “incomplete”

