



PRODUCT RELEASE ANNOUNCEMENT

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Product Group	CATIA V5 for NX
Release Version	21.0

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History

Revision	Update Information
1.0	Initial Release

Product Codes

Product Code	Product Description
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) 3D PDF Add On
CA5NX-A2	CATIA V5 (CAA Based) to > NX (uni-directional) PMI Add On
CA5NX-A3	CATIA V5 (CAA Based) to <> NX (bi-directional) PMI Add On
NXCA5-U1	NX to CATIA V5 (CAA Based) (uni-directional)
NXCA5-A1	NX to CATIA V5 (CAA Based) (uni-directional) PMI Add On
NXCA5-A2	NX to CATIA V5 (CAA Based) (uni-directional) 3D PDF Add On
CATNX-U1	CATIA V4 > NX
CATNX-U3	CATIA V4 > NX (with 2D drawing support)
CATNX-B1	CATIA V4 <> NX
CATNX-B3	CATIA V4 <> NX (with 2D drawing support)



Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V4	All revisions
CATIA V5	CATIA V5-6R2015 , CATIA V5-6R2016 & CATIA V5-6R2017
NX	NX 12

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Window 8 64bit	Tested and but not fully supported as Siemens have stopped support for this OS at NX 12
Window 10 64bit	Not formally qualified for CATIA V5-6R2015 Qualified and fully supported for CATIA V4, V5-6R2016 & V5-62017.

Distribution Media

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

Product	URL for CD image download
Translator Media	CAD 21.0 CA5NX12 WIN.01
Unified Interface	CAD 21.0 UI WIN.01
License Server	CAD FLEXLM 11.14 WIN.01

New Features / Enhancements

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

Product Impacted	New Feature / Enhancement Description
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CATIA V5 <> NX	Interim release for the Support of NX 12 to V5 R25, R26, R27
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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>UI:</i> Translation will fail if defined output path in “translator options” does not exist. Please ensure you define a valid existing output path.
2	<i>NX Interactive:</i> V5 CATPart / CATProduct supported using the interactive import / export method. CGR / V4 are not supported this way.
3	<i>CATIA V5 PMI Read:</i> Some data positioning errors when reading data from CATIA V5 R27 – Error manifests in log as “Error: While retrieving GDI attributes”. Awaiting resolution from Dassault.
4	<i>NX Unique Part Naming:</i> In CATIA V5, file name uniqueness includes the file extension so you could have an assembly that contained both XXXX.CATProduct and XXXX.CATPart. Therefore as the basename for both these are the same NX would produce a request for XXXX.prt twice and a corrupt assembly would be produced. In an attempt to avoid such a saturation we currently leave the extension on if it’s a CATProduct thus ensuring uniqueness of the output filename and avoiding an assembly corruption.
5	<i>Instance Colour Processing NX > V5:</i> By default the NX to V5 translator will not process occurrence / instance colours from NX. Setting TS_NXCATIA5_USE_INSTANCE_COLOURS will allow this



Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID	Brief Description

