PRODUCT RELEASE ANNOUNCEMENT

Product Category	**CADverter**
Product Group	CATIA V5 for NX
Release Version	21.2

Document Type	**Product Release Announcement**
Status	Released
Revision	2.0
Author	Product Manager
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History

Revision	Update Information
1.0	Initial Release
2.0	Added 3rd party NX limitations

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 & CATIA V5-6R2018
NX	NX 10, NX 11, NX 12

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Window 7 64bit	Tested but not fully supported as Siemens have stopped support for this OS at NX 12
Window 8 64bit	Tested 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 & V5-62018

Distribution Media

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

Product	URL for CD image download
Translator Media	CAD_21.2_CA5NX_WIN.01
Unified Interface	CAD_21.2_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

Ref ID	New Feature / Enhancement Description
1	Support for CATIA V5 R28
2	Improved processing of Instance Colours
3	Improved processing of PMI view associations
4	Interactive processing enhanced to allow importing of cgr files into NX.
5	Support for processing NX containing subordinate JT files
6	Enhanced UI to allow for configurable attribute processing – V5 to NX

Known Limitations and Restrictions

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

Ref ID	Limitations and Restrictions
1	*UI*: Translation will fail if defined output path in "translator options" does not exist. Please ensure you define a valid existing output path.
2	*NX Interactive:* V5 CATPart / CATProduct / cgr supported using the interactive import / export method. Capitalised CGR are not currently supported this way.
3	For Windows 10 - NX 11 users experiencing issues with the error message: "Standard C++ Exception Caught" the version of NX will need upgrading to 11.0.2 with MP10. The issue was reported by Siemens to have been caused by an automatic Windows update. Upgrade to the above version to resolve.
4	Siemens introduced an error at version 12.0.1 (Siemens PR9093785) that caused a failure. Error message reporting issue with: libnxopencpp.dll. to resolve this use the following NX versions: • 12.0.1.* with MP1 (maintenance Pack 1) • 12.0.2.*

Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID	Brief Description
CAS-03495-F1S8D6	Subordinate JT files not translated
CAS-03564-F5R1C7	Incorrect PMI positioning
CAS-03580-B7K5G6	CATIA V5 attributes not processed