PRODUCT RELEASE ANNOUNCEMENT

Product Category	CADverter
Product Group	CATIA V5 for NX
Release Version	23.0

Document Type	Product Release Announcement
Status	Released
Revision	1.0
Author	Product Manager
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Contents

History

Revision	Update Information
1.0	Initial Release

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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 V5-6R2017, CATIA V56R2018, CATIA V5-6R2019 & CATIA V5R2020
NX	NX 11, NX 12, NX 1847, NX 1872 & NX 1899

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	Qualified and fully supported for CATIA V4, V5-62017, V5-62018, V5-62019 & CATIA V5R2020

Distribution Media

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

Product	URL for CD image download
Translator Media	CAD_23.0_CA5NX_WIN.01
Unified Interface	CAD_23.0_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

Ref ID	New Feature / Enhancement Description
1	Improvements in maintaining visibility state of geometry
2	Support for processing NX parts that contain both Solid & Tessellated geometry to separate CATIAV5 CATPart for BREP & CGR file for Tessellated geometry
3	Improved warning message when importing CATIA into NX
4	Ability to read NX Scanned Data
5	Support for NX 1872 & 1899
6	Support for CATIA V5-62020
7	Support for interactive CAD upload to Theorem XR

Known Limitations and Restrictions

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

Ref ID	Limitations and Restrictions
1	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.
2	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