# PRODUCT RELEASE ANNOUNCEMENT

| Product Category | CADverter |
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| Product Group | CATIA V5 for NX |
| Release Version | 20.2 |

| Document Type | Product Release Announcement |
| --- | --- |
| Status | Released |
| Revision | 2.0 |
| Author | Product Manager |
| Issued | 05/01/2018 |

## Contents

## History

| Revision | Update Information |
|---|---|
| **1.0** | Initial Release |
| **2.0** | Update to include V4 Product codes |

## 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 R19, R20, R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014, CATIA V5-6R2015 , CATIA V5-6R2016 & CATIA V5-6R2017 |
| **NX** | NX 9.0, NX 10, NX 11 |

## Supported Operating Systems

The following operating systems have been qualified with this release

| Operating System | Qualification Status |
|---|---|
| **Window 8 64bit** | Not formally qualified for CATIA V5 R19, R21 and CATIA V5-6R2012<br><br>Qualified and fully supported for CATIA V5-6R2013, CATIA V5-6R2014 CATIA V5-6R2015 & CATIA V5-6R2016 Only |
| **Window 10 64bit** | Not formally qualified for CATIA V5 R19, CATIA V5 R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014 & CATIA V5-6R2015<br><br>Qualified and fully supported for CATIA V4, V5-6R2016 & V5-62017. |
| **Windows Server 2008 R2** | Not formally qualified for CATIA V5 R19<br><br>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 2012** | Not formally qualified for CATIA V5 R19, CATIA V5 R21, CATIA V5-6R2012 and CATIA V5-6R2013<br><br>Qualified and fully supported for CATIA V5-6R2014. CATIA V5-6R2015 & CATIA V5-6R2016 Only |

## Distribution Media

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

| Product | URL for CD image download |
| --- | --- |
| **Translator Media** | CAD_20.2_CA5NX_WIN.01 |
| **Unified Interface** | CAD_20.2_UI_WIN.03 |
| **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 |
|---|---|
| **CATIA V5 <> NX** | Support for CATIA V5-6R2017 has been implemented with this release |
| **CATIA V5 <> NX** | *Installer update:* Theorem have transitioned to a standard Microsoft Installer (.msi) package |
| **CATIA V5 <> NX** | *Model Views:* Allowing the processing of Capture / Model views |
| **CATIA V5 to NX** | Enhanced PMI reading |
| **NX to CATIA V5** | Included new environment option "TS_NXCATIA5_USE_INSTANCE_COLOURS" that will allow instance colours to be processed. |
| **NX to CATIA V5** | Interactive import / export menus available for users |
| **NX to CATIA V5** | Added controls for processing of Sliver faces. |
| **NX to CATIA V5** | *Assembly Arrangements:* Support for reading NX Assembly arrangements |
| **NX to CATIA V5** | *Face / Body Translucency:* Enhancements to translucency processing |
| **NX to CATIA V5** | *Log file updates:* Sanitizing error messages to be more concise |

## 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 supported using the interactive import / export method. CGR / V4 are not supported this way. |
| 3 | *CATIA V5 PMI Read:* 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 | *NX Unique Part Naming*: 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 | *Instance Colour Processing NX > V5*: 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 |
| --- | --- |
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