PRODUCT RELEASE ANNOUNCEMENT

Product Category	**CADverter**
Product Group	CATIA V5 <> CREO
Release Version	21.3

Document Type	**Product Release Announcement**
Status	Released
Revision	1.0
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Issued	13/11/2018

Contents

History

Revision	Update Information
1.0	Initial Release.

Product Codes

Product Code	Product Description
CA5CRE-B1	CATIA V5 <> CREO Bi-Directional Translator
CA5CRE-U1	CATIA V5 > CREO Uni-Directional Translator
CRECA5-U1	CREO > CATIA V5 Uni-Directional Translator
CA5CRE-A1	CATIA V5 <> CREO PMI Add-on Module
CA5CRE-A2	CATIA V5 > CREO PMI Add-on Module
CA5CRE-A3	CATIA V5 > Creo PDF Add-on Module
CRECA5-A1	CREO > CATIA V5 PMI Add-on Module

Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
Creo Parametric	3.0 (M040 or Higher is recommended) 4.0 (M030 or Higher is recommended) 5.0
CATIA V5	CATIA V5-6R2016 CATIA V5-6R2017 CATIA V5-6R2018

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Windows 7 64bit	Qualified and fully supported
Windows 8 64bit	Qualified and fully supported
Windows 10 64 Bit	Qualified and fully supported

Distribution Media

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

Software	URL for CD image download
Installation Media	CAD_21.3_CA5CRE_WIN.01
Unified Interface	CAD_21.3_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

Creo <> CATIA V5

Ref ID	New Feature / Enhancement Description
1	Support for CATIA V5-6R2017 & CATIAV5-6R2018
2	Support for CREO Parametric 4.0 & Creo Parametric 5.0
3	Bi-Directional Attribute support added
4	Added support for the chop_name argument. This is used to assist in making unique filenames when filenames exceed the 30 character limit imposed by Creo. (default is 20 characters)
5	UI Creo Read Instance argument added
6	Support for CREO seed parts and assemblies

Known Limitations and Restrictions

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

Creo <> CATIA V5

Ref ID	Limitations and Restrictions
1	Some Assembly instance colours read from Creo will not show.
2	Creo Interactive imports & exports will not show the Configuration description when changed.
3	Creo Interactive exports to CATIA V5 will save .CATProduct in the file name by default.

Resolved Support Calls

The following list of Support Calls are resolved with this release

Creo <> CATIA V5

No outstanding support calls resolved at this release.