PRODUCT RELEASE ANNOUNCEMENT

Product Category	CADverter
Product Group	CATIA V5 to Creo View for Windchill
Release Version	25.2

Document Type	Product Release Announcement
Status	Released
Revision	3.0
Author	Product Manager
Issued	19-Jul-22

Contents

History

Revision	Update Information
1.0	Draft
2.0	Review
3.0	Release

Product Codes

Product Code	Product Description
CA5CVW-U1	CATIA V5 to Creo View
CA5CVW-U2	CATIA V5 to Creo View for Windchill
CA5CVW-S1	CATIA V5 to Creo View - PMI Add On
CA5CVW-S2	CATIA V5 to Creo View for Windchill Extended CGR
CA5CVW-S3	CATIA V5 to Creo View for Windchill Post Processing Module
CA5CVW-S4	CATIA V5 to Creo View - JT Add On
CA5CVW-S5	CATIA V5 to Creo View - 3D PDF Add On
CA5CVW-S6	CATIA V5 to Creo View - Composites Add On

Windchill / Creo View - CATIA Compatibility Matrices

The following is a link to the PTC support pages (only accessible with a valid PTC customer login). This contains a matrix of supported software configurations. The information found by this link is subject to change and remains the property of PTC.

Windchill / Creo View - CATIA Compatibility Matrices

Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V5	V5-6R2019, V5-6R2020, V5-6R2021 & V5-6R2022
Creo View Client	Up to Creo View 8.1

Supporting API's

The following supporting API versions are used

Application	Revision
JTOpen	JT10.8.0
3D PDF	HOOPS Exchange Publish 2020 (V13.0)
Creo View DAPI	8.1.0.0 - (18.1.0.0)

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Windows 10 64bit	Qualified and fully supported for CATIA V5-6R2019, V5-6R2020, V5-6R2021 & V5-6R2022

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Distribution Media

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

Product	URL for CD image download
ALL currently supported Versions of CATIA V5	Product Download Information Flex License Server Install 11.17 Theorem UI Installer 25.1 This product has its own release cycle, for the latest update visit Documentation & Product Release Notes and select 'Unified Interface'

New Features / Enhancements

The following new features or enhancements are introduced with the 25.2 release.

Ref ID	New and Changed
1	CGR Structures are now displayed in both Standard and Minimum Memory mode
2	Component ID is ON by default
3	Enhancements to Material properties on multi-body components
4	Ability to render with/without material colour added to recipe
5	Support for CVA Toolkit (dAPI) 8.1
6	Support for CATIA V5-6R2022 (R32)

Known Limitations and Restrictions

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

Ref ID	Limitations and Restrictions
1	Certain features are not available when using Minimum Memory. It is recommended that Standard processing is used when precise conversion of PMI and captures are required. Standard versus Minimum Memory processing is detailed in the user documentation.
2	It has been found that running worker daemon from the default install path C:\Program Files\Theorem causes the daemon to fail. The recommendation is to install the CADverter in a path other than the default which contains no spaces, if this is required.
3	Using Standard mode when translating very large assemblies with multi-body components can consume lots of memory and can cause the translation to fail. Turning off multi-body or switching to Minimum Memory mode in these cases is recommended.
4	Certain additional formats are no longer supported at CATIAV5R29. These include Navrep, VPS and 3dmap. Therefore they are no longer available in R30, R31, R32. A full review of the supported formats will be carried out at the next revision.

Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID	Brief Description
CAS-04690	Representation of CGR files are now created in 'Minimum Memory' Conversion Mode.
CAS-04733	Theorem 24.2 - Face colors which were overwritten by the material colour can be reverted to their original behaviour by switching off material rendering.
CAS-04744	PTC_WM_PART_NUMBER/NAME parameter is now visible in Creo View for sub-level CAD Assemblies
CAS-04769	Some CATParts which consisted of many faces were not translated in Standard mode. Changing to Minimum memory mode or switching off multi-body solves this problem in most cases
CAS-04784	Mass property results which regressed in certain cases when material read was added are now correctly written
CAS-04805	STEP files which were not being created by specific CATP* which were named in the form yxz..CATPart, abc..CATProduct are now processed correctly.
CAS-04806	WorkerDaemon.exe will not run from default installation folder. If this needs to be run standalone you will need to move it to a folder which has no spaces in the path
CAS-04823	Volume values which were missing in Creo View are now properly assigned
CAS-04828	Issue with multiple surface CATPart processing has been fixed