THEOREM
SOLUTIONS

PRODUCT RELEASE ANNOUNCEMENT

Product Category	CADverter
Product Group	CATIA V5 to Creo View for Windchill
Release Version	27.1

Document Type	Product Release Announcement
Status	Released
Revision	1.0
Author	Product Manager
Issued	12-Aug-24

Contents

History

Revision	Update Information
01	In Work

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	V5-6R2020, V5-6R2021, V5-6R2022 & V5-6R2023, V5-6R2024
Creo View Client	Up to Creo View 11.0

Supporting API's

The following supporting API versions are used

Application	Revision
JTOpen	JTTK 11.4.0
3D PDF	HOOPS Exchange Publish 2023
Creo View DAPI	20.1.0.0

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 V5-6R2020, V5-6R2021, V5-6R2022, V5-6R2023 & V5-6R2024
Windows 11	This release has been tested and is supported on Windows 11 with the following caveats. • CATIA V5-6R2024 (R34) is supported on Windows 11, earlier versions of CATIA may not be fully supported on Windows 11. Please contact your supplier for full support details.

	Creo View versions 9.1 and above are supported on Windows 11, earlier versions of Creo View may not be fully supported on Windows 11. Please contact your supplier for full support details.

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 11.19* Theorem UI Installer 27.0 This product has its own release cycle, for the latest update visit Documentation & Product Release Notes and select 'Unified Interface'

∗ This FlexLM version is supported on Windows 10 & 11 and needs an updated license file.

New Features / Enhancements

The following new features or enhancements are introduced with the 26.3 release.

Ref ID	New and changed
1	Support for CATIA V5-6R2024.
2	Support for Monochrome PDF output (requires a copy of Ghostscript which is not supplied by Theorem)
3	
4	
5	
6	

Known Limitations and Restrictions

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

Ref ID	Limitations and Restrictions
1	The use of colour and transparency on are not currently supported when defined in different levels on an instance in an assembly. E.g. If the colour is defined on a body and the transparency is defined on the instance then the colour only will be shown.
2	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.
3	The trouble-shooting section has been removed from the User Guide and will appear as an updated separate document shortly after this release.

Resolved Support Calls

The following list of Support Calls are resolved with this release

Theorem Call ID/ PTC ID	Brief Description
REQ-893	It is now possible to convert PDF drawing in monochrome. See user guide for details and dependencies.
14559561	Unit Scaling issue for Inertia metrics is now correct
TQA-28/ 15146234	Mirrored geometry should now be converted correctly
TH-293/ 16583524	Color render issue between the 24.2 & 25.2 on File Sync Worker mode
TH-417/TQA-21/ 16721090	Moments Of Inertia from CATIA data are now correct in Creo View.
TH-557/TQA-4/ 16721090	Objects should now be rendered correctly
TH-609	Creo view publishing dimensions with fill in wrong orientation
TH-627	Theorem CADVerter long time needed to generate "thumbnail"
TH-792/ 17262181	if adapter/outputSingleShape is enabled then Views & Captures (PMI) will not be published. This is a current restriction.
REQ-1100/ 15146072	Minimum Memory mode will be deprecated at a future Revision