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## PRODUCT RELEASE ANNOUNCEMENT

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Product Category	<b>CADverter</b>
Product Group	CATIA V5 to Creo View for Windchill
Release Version	27.2

Document Type	<b>Product Release Announcement</b>
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Revision	1.0
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## History

Revision	Update Information
01	In Work



## Product Codes

Product Code	Product Description
<b>CA5CVW-U1</b>	CATIA V5 to Creo View
<b>CA5CVW-U2</b>	CATIA V5 to Creo View for Windchill
<b>CA5CVW-S1</b>	CATIA V5 to Creo View - PMI Add On
<b>CA5CVW-S2</b>	CATIA V5 to Creo View for Windchill Extended CGR
<b>CA5CVW-S3</b>	CATIA V5 to Creo View for Windchill Post Processing Module
<b>CA5CVW-S4</b>	CATIA V5 to Creo View - JT Add On
<b>CA5CVW-S5</b>	CATIA V5 to Creo View - 3D PDF Add On
<b>CA5CVW-S6</b>	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
<b>CATIA V5</b>	V5-6R2020, V5-6R2021, V5-6R2022 & V5-6R2023, V5-6R2024
<b>Creo View Client</b>	Up to Creo View 11.0

## Supporting API's

The following supporting API versions are used

Application	Revision
<b>JTOpen</b>	JTK 11.4.0
<b>3D PDF</b>	HOOPS Exchange Publish 2023
<b>Creo View DAPI</b>	20.1.0.0

## Supported Operating Systems

The following operating systems have been qualified with this release.

Operating System	Qualification Status
<b>Windows 10 64bit</b>	Qualified and fully supported for CATIA V5-6R2020, V5-6R2021, V5-6R2022, V5-6R2023 & V5-6R2024  <u>NOTE! Windows 10 is out of support in 2025 and</u>
<b>Windows 11</b>	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
<b>ALL currently supported Versions of CATIA V5</b>	<a href="#">Product Download Information</a> <a href="#">Flex License Server Install 11.19*</a> <a href="#">Theorem UI Installer 27.0</a> This product has its own release cycle, for the latest update visit <a href="#">Documentation &amp; Product Release Notes</a> 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**

<b>1</b>	Support for CATIA V5-6R2020 thro' V5-6R2024.
<b>2</b>	
<b>3</b>	
<b>4</b>	
<b>5</b>	
<b>6</b>	





## Known Limitations and Restrictions

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

**Ref ID    Limitations and Restrictions**

<b>1</b>	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.
<b>2</b>	At this revision 'Minimum Memory mode is deprecated. It is targeted to be removed during the 28.x timescale (year2025).



## Resolved Support Calls

The following list of Support Calls are resolved with this release

<b>Theorem Call ID/ PTC ID</b>	<b>Brief Description</b>
<b>TH-1021</b>	If the Center of Gravity value is positive, then its conversion occurs in meters, if the value is negative, then the conversion occurs in millimeters
<b>TH-1023</b>	The saved representation does not have an additional directory with separate JT files for the CAD parts during JT conversion
<b>TH-1018</b>	The .pvs file structure is incorrect if the subproduct name and component node name are identical
<b>TH-417</b>	Moments Of Inertia from CATIA data are not correct in Creo View. Note - Was previous reported as fix - it has been re-worked for this release
<b>TH-1020</b>	Incorrect conversion of the CATIA V5 part containing a Mirror feature in the Design Tree. This same data has been raised before (CON-3492: QA : Solid with mirrored body does not tessellate correctly in Creo ViewComplete ) and commented on in previous test notes
<b>TH-1022</b>	Incorrect feature color conversion. The setting adapter/read_face_colour=1 needed, this used to be the default but was found to be more efficient if turned off as default.
<b>REQ-1293/TH-990</b>	The command -d1 can now be added to the recipe file as an advanced option. This displays the data from the upper node, similar to the command line option.

