



PRODUCT RELEASE ANNOUNCEMENT

Product Category	Visualize 3D
Product Group	CATIA V5 for CREO
Release Version	19.3

Document Type	Product Release Announcement
Status	Released
Revision	1.0
Author	Product Manager
Issued	10/08/2016



Contents

History	2
Product Codes	3
Supported Application Revisions.....	4
Supporting API's	4
Supported Operating Systems.....	4
Distribution Media.....	5
New Features / Enhancements	6
Known Limitations and Restrictions	7
Resolved Support Calls	7



History

Revision	Update Information
1.0	Initial Release

Product Codes

Product Code	Product Description
CA5CRE-B1	CATIA V5 (CAA Based) <> Creo Parametric Bi-Directional Base Product
CA5CRE-A1	CATIA V5 (CAA Based) <> Creo Parametric Add-on module for PMI
CA5CRE-U1	CATIA V5 (CAA Based) > Creo Parametric Uni-Directional Base Product
CA5CRE-A2	CATIA V5 (CAA Based) > Creo Parametric Add-on module for PMI
CRECA5-U1	Creo Parametric > CATIA V5 (CAA Based) Uni-Directional Base Product
CRECA5-A1	Creo Parametric > CATIA V5 (CAA Based) Add-on module for PMI

Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V5	R19, R20, R21, V5-6R2012, V5-6R2013, V5-6R2014, V5-6R2015, V5-6R2016
Creo Parametric	2.0, 3.0

Supporting API's

The following supporting API support is used

Application	Revision
N/A	

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Windows 7 64bit	CATIA and Creo - Qualified and fully supported all versions
Window 8 64bit	CATIA: <ul style="list-style-type: none"> • Not formally qualified for CATIA V5 R19, R20, R21, V5-6R2012 • Qualified and fully supported for V5-6R2013, V5-6R2014, V5-6R2015, V5-6R2016 Creo: 2.0, 3.0
Windows Server 2008 R2	CATIA: <ul style="list-style-type: none"> • Not formally qualified for V5 R19, R20 • Qualified and fully supported for R21, V5-6R2012, V5-6R2013, V5-6R2014, V5-6R2015, V5-6R2016 Creo: <ul style="list-style-type: none"> • Qualified and fully supported for 2.0 • Not formally qualified for 3.0
Windows Server 2012	CATIA: <ul style="list-style-type: none"> • Not formally qualified for V5 R19, R20, R21, V5-6R2012, V5-6R2013



	<ul style="list-style-type: none"> • Qualified and fully supported for V5-6R2014, V5-6R2015, V5-6R2016 <p>Creo:</p> <ul style="list-style-type: none"> • Qualified and fully supported for 2.0 M150 or later and 3.0
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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
Support for ALL Versions	cdCAD 19.3 CA5CRE WIN.01.zip
Unified Interface	cdCAD 19.3 UI WIN.01.zip



New Features / Enhancements

The following new features or enhancements have been introduced with this release

Ref ID New Feature / Enhancement Description

1	<p><i>New Product :</i></p> <p>This is the first release of a new bi-directional translator between CATIA V5 (CAA) and Creo Parametric.</p> <p>The following features are supported in this new product offering:-</p> <ul style="list-style-type: none"> • Standard Assembly Processing • Standard Geometry Processing: <ul style="list-style-type: none"> ○ BREP Solids ○ Analytic Surfaces / Curves ○ Nurbs Surfaces / Curves ○ Wireframe ○ CGR • Document Properties • Material Properties (Colour, Transparency, etc) • Mass Properties • PMI <p>Large Assembly Processing</p>
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Known Limitations and Restrictions

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

Ref ID Limitations and Restrictions

1	<p><i>Creo has a filename length restriction of 32 characters. When translating from CATIA V5 to Creo, if the CATIA filename exceeds this limit, the filename will be shortened and appended with a unique identifier.</i></p> <p><i>If translating using “Large Assembly Processing”, there is an issue with the filename lengths.</i></p>
2	<p><i>When translating CATIA to Creo with PMI, the CATIA Captures will be converted into Creo Combined States. Creo does not allow special characters in the names of Combined States, such as @, #, \$, &, etc.. This will not cause a translation to fail, but the Combined State will not be created.</i></p>

Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Brief Description
Call ID

N/A	
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