PRODUCT RELEASE ANNOUNCEMENT

Product Category	**Visualize 3D**
Product Group	CATIA V5 for JT
Release Version	19.4

Document Type	**Product Release Announcement**
Status	Released
Revision	1.0
Author	Product Manager
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Contents

History

Revision	Update Information
1.0	Initial Release

Product Codes

Product Code	Product Description
CA5JT-U1	CATIA V5 (CAA Based) to JT (uni-directional)
JTCA5-U1	JT to CATIA V5 (CAA Based) (uni-directional)
CA5JT-A5	CATIA V5 (CAA Based) to JT (uni-directional) with PMI Add On
CA5JT-A6	CATIA V5 (CAA Based) to JT (uni-directional) with Electrical Add On
CA5JT-A7	CATIA V5 (CAA Based) to JT (uni-directional) with 3D PDF Add On
JTCA5-A1	JT to CATIA V5 (CAA Based) (uni-directional) with 3DPDF Add On
JTCA5-A2	JT to CATIA V5 (CAA Based) (uni-directional) with 3DXML Export Module
CAGJT-U1	CATIA V4 to JT (uni-directional)

Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V5	CATIA V5 R19, R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014, CATIA V5-6R2015 & CATIA V5-6R2016
JTOpen	JTK 8.0.0.0
Adobe Reader	Adobe Reader XI

Supporting API's

The following supporting API support is used

Application	Revision
3D PDF API	HOOPS Exchange Publish 8.0

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Windows 7 64bit	Qualified and fully supported (All CATIA V5 Revisions)
Window 8 64bit	Qualified and fully supported for CATIA V5-6R2013, CATIA V5-6R2014 , CATIA V5-6R2015 & CATIA V5-6R2016
Windows Server 2008 R2	Qualified and fully supported for CATIA V5 R21, CATIA V5-6R2012, CATIA V5-6R2013, CATIA V5-6R2014, CATIA V5-6R2015 & CATIA V5-6R2016
Windows Server 2012	Qualified and fully supported for CATIA V5-6R2014 , CATIA V5-6R2015 & CATIA V5-6R2016

Distribution Media

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

Product	Select link to download CD images
CATIA V5-6 (CAA) – JT installation media	[Download Version 19.4 CATIA V5 (CAA) - JT CD image](#)
Theorem Unified Interface installation media	[Download Version 19.4 Unified Interface CD image](#)

New Features / Enhancements

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

Product Impacted	New Feature / Enhancement Description
Theorem Interactive Export/Import Menus	New support for Unified Interface configurations under the 'Configure' tab.
JT to CATIA V5-6 **CATIA V5-6 to JT**	New Add On licensing functionality for 3DPDF (licensed separately).
CATIA V5-6 to JT	TEXT_NAME - Names text based on first few characters as per V5 spec tree
CATIA V5-6 to JT	READ_CGR_EDGES & DONT_READ_CGR_EDGES to control reading edge curves from CGR files
CATIA V5-6 to JT	READ_ASSY_PMI now takes optional 2nd argument n where n is a int from 1 to 63 and is the number of assembly levels from which to read PMI
CATIA V5-6 to JT	Added FSOL_VERTEX_NORMAL, FSOL_FACET_NORMAL & FSOL2 to control type of facetted data read and passed to write leg
CATIA V5-6 to JT	The translator now only requires a single Daimler configuration file. Previously required one for Parts and one for Assemblies.
CATIA V5-6 to JT	Change to mass property attributes for Daimler Compliance. The property CAD_MASS_UNITS is now always added regardless of whether CAD_MASS has been added. CAD_DENSITY is only added if it has been set by the read leg, or if the complete set of mass properties have been calculated.
CATIA V5-6 to JT	Added a new completion message:- "Complete XX File created but some solids may be open" and implemented its use in the JT write leg for XTBrep. The message indicates that there have been problems creating all the solids, but the subsequent creation of faces has been completely successful. Also removed some unnecessary messages from the JT write output. (REQ00195)
JT to CATIA V5-6	REMOVE_TOP_GROUP / DONT_REMOVE_TOP_GROUPS Controls if groups are passed to CATIA V5 whether first group is kept or not

JT to CATIA V5-6	Correction to PMI association read. Code to ensure that associated edge and face ids were unique in an assembly context, wasn't working correctly.
JT to CATIA V5-6	Option to tessellate BREP data directly, rather than by reading existing LOD data. The output is stored in the GCO FSOL2 entity to enable it to be written for the STEP write leg.
JT to CATIA V5-6	Modifications to the reading of SUBNODE data to improve efficiency when there are a large number of nodes. Also the option "spec_tree" has been added to store the subnode structure so that write legs can create a specification tree.
JT to CATIA V5-6	When an error is returned from the PLMXML API when loading a PLMXML file, attempt to continue rather than terminating. Even when an error is returned, the XML may have been loaded.
JT to CATIA V5-6	During processing of a PLMXML assembly, each JT file referenced by the PLMXML file is read and processed. Previously, if there was a clash of naming between two of the JT files, then an error was reported by the toolkit. This was caused by the processed file not being completely removed from the translator memory before the next file was processed. This has been corrected.
JT to CATIA V5-6	Allow for nodes with more than one transform attached. An assembly stored in metres and containing millimetre parts contained two transforms on the nodes referencing the millimetre parts. The first positioned the node and the second scaled the node. This modification combines all the transforms on each node so will also cope with more complicated combinations.
JT to CATIA V5-6	Colours are now not written to individual facets in an Fsol if the JT shape or part has colour, as the colour on the part or shape normally overrides the facet colours.

Known Limitations and Restrictions

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

Ref ID	Limitations and Restrictions
1	*CATIA V5 Viewed Data:* CATIA V5 data will not include PMI in the Unified Interface
2	*PDF Add On Output:* When running an *Interactive* 'Save As' output to PDF as part of the Add On product, PMI will be output without a V5>JT PMI license. To enable this through the UI or in Batch the user must buy a PMI license or a full V5>3DPDF *On Demand* product.
3	*Interactive V5 Theorem Export to JT:* CATIA V5 will throw a 'File Not Found' error when outputting to PLMXML data due to the Application expecting and checking for a JT file. The data will still be created successfully.
4	*Interactive V5 Theorem Import to CGR:* CATIA V5 will throw a 'File Not Found' error when importing JT to CGR format using CGR as the output type within a configuration but CATPart selected from the Interactive menu.

Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID	Brief Description
CAS-02725	Successful translation with Daimler configuration file.
CAS-02729	Missing geometry in output.
CAS-02759	Crash when using Daimler configuration in German locale
CAS-02820	Output data is Daimler compliant
CAS-02827	Incorrect surface geometry
CAS-02933	Interactive export menu crash
CAS-02975	Surface errors in output data