PRODUCT RELEASE ANNOUNCEMENT

Product Category	**CADverter**
Product Group	JT for CATIA V5 Multi-CAD
Release Versions	19.3

Document Type	**Product Release Announcement**
Status	Released
Revision	1.0
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Issued	31/08/2016

Contents

History

Revision	Update Information
1.0	Initial Release

Product Codes

Product Code	Product Type	Product Description
MC5JT-B1	Base Product	CATIA V5 (Multi-CAD) to/from JT (bi-directional)
MC5JT-U1	Base Product	CATIA V5 (Multi-CAD) to JT (uni-directional)
JTMC5-U1	Base Product	JT to CATIA V5 (Multi-CAD) (uni-directional)
MC5JT-A1	Optional add-on	CATIA V5 (Multi-CAD) to JT (uni-directional) Model Based Definition (3D Dimensions & Annotations)
JTMC5-A1	Optional add-on	JT to CATIA V5 (Multi-CAD) (uni-directional) Model Based Definition (3D Dimensions & Annotations)

Supported Application Revisions

The following application revisions have been qualified with this release

Application	Revision
CATIA V5	CATIA V5-6R2013 (SP4 minimum recommended) CATIA V5-6R2014 (SP4 minimum recommended) CATIA V5-6R2015 CATIA V5-6R2016

Supporting API's

The following supporting API support is used

Application	Revision
Dassault Multi-CAD (XCAD)	CATIA V5-6R2013 CATIA V5-6R2014 CATIA V5-6R2015 CATIA V5-6R2016
JTOpen Libraries	JTK 8.0.0.0

Supported Operating Systems

The following operating systems have been qualified with this release

Operating System	Qualification Status
Windows 7 64bit	Qualified and fully supported
Window 8 64bit	Qualified and fully supported for CATIA V5-6R2013, CATIA V5-6R2014 CATIA V5-6R2015 and CATIA V5-6R2016
Windows Server 2008 R2	Qualified and fully supported for CATIA V5-6R2013, CATIA V5-6R2014 CATIA V5-6R2015 and CATIA V5-6R2016
Windows Server 2012	Not formally qualified for CATIA V5-6R2013 Qualified and fully supported for CATIA V5-6R2014 CATIA V5-6R2015 and CATIA V5-6R2016 only

Distribution Media

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

Note! For each CATIA V5 release the CATIA V5 Multi-CAD software requires the relevant Dassault Systems TXX media to be downloaded and installed prior to installing the Theorem Multi-CAD Product Software. Please see the Software Installation guide for further details.

Product	Select hyperlink to download installation CD
TXX for CATIA V5-6R2013	Download TXX Platform Media for CATIA V5-6R2013
TXX for CATIA V5-6R2014	Download TXX Platform Media for CATIA V5-6R2014
TXX for CATIA V5-6R2015	Download TXX Platform Media for CATIA V5-6R2015
TXX for CATIA V5-6R2016	Download TXX Platform Media for CATIA V5-6R2016
Multi-CAD CATIA V5-6 – JT for all supported CATIA V5 releases	Download CAD_19.3_MC5JT_WIN.01 Release
Generic Theorem User Interface	Download CAD_19.3_UI_WIN.02 Release

New Features / Enhancements

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

Product Impacted	New Feature / Enhancement Description
JT to CATIA V5-6	Additional JT utilities provided to analyse the content of a JT file. These are **jt_content.cmd, jt_layers.cmd** and **jt_validation.cmd**. See separate documents "*JT File Analysis Utilities User Guide*" and "*JT File Validation Utility*" for a complete description of functionality available.
JT to CATIA V5-6	Colours are now not written to individual facets in a Faceted Solid if the JT shape or part has colour, as the colour on the part or shape normally overrides the facet colours.
JT to CATIA V5-6	Addition of the "**detail_name**" advanced command line option. When a detail name string is specified, it is used, along with the gco_id of the detail to create a short (and meaningless) name for the detail, for systems that can't cope with long names.
JT to CATIA V5-6	Increase the GCO entity count used to handle PMI processing when each polyline of the PMI graphics is stored as a separate pcurve.
JT to CATIA V5-6	Prevent invalid solids (he_id = -1) from having a usage flag of "normal". This was causing the V5 write problems when they are part of a detail.
JT to CATIA V5-6	Fix XCAD server related problem where some static storage was not reset for a subsequent call of a function
JT to CATIA V5-6	Change to optionally store PMI strokes as pcurves to allow for non-planar PMI. Advanced command line option: **pmi_pcurves**.
JT to CATIA V5-6	Change to prevent negative PMI ids from being used as GCO HEID's, causing the write leg to ignore.
JT to CATIA V5-6	Cope with the situation where there is geometry at the sub-assembly level when creating a plmxml file that references "sub-node" JT files. Previously, the plmxml files referenced JT parts below the sub-node level that did not exist. (N.B. There was also a bug in the V5 read leg that left geometry at the sub-assembly level, so when this is corrected, the test case will not demonstrate the problem.)
JT to CATIA V5-6	Added option handling for pmi_filter_file option. Tools Option page modified and option passed to jt_vwr command line.
JT to CATIA V5-6	Add check for invalid SREV generator pcurves discovered in JT sample.
JT to CATIA V5-6	Add cascade of BREP assigned colour down to faces in case XCAD import of BREP fails, and individual faces are created without parent BREP colour attributes.

JT to CATIA V5-6	Fixed issue for V5 "Insert Existing Component" - failing to read ANNOTATIONS_3D=TRUE in the CATIA JtToNavConfigs.txt file.
JT to CATIA V5-6	Add _PMI option license check and handling in TSCCATReader module.
JT to CATIA V5-6	Add handling of PMI associative links to geometry.
JT to CATIA V5-6	Improve fixup for JT_BREP models by applying simplify_solids settings if no surfaces with attribute PSSURFACE are found in the part, indicating it is not a XT_BREP model. Also include environment variable check of TS_IGNORE_JT_BREP_SIMPLIFY_FIXUP to permit the modification to be ignored.
JT to CATIA V5-6	Change to the code that determines the "up" vector for a view to line up with the latest Siemens documentation.
JT to CATIA V5-6	The setting of a default colour from the shapes within a part has been modified to only use the shape colour if it is a tri-strip shape. Previously all shapes were looped through and in one case this resulted in a GCO BREP and structure colour getting set from a coloured line.
CATIA V5-6 to JT	Modifications to PMI associativity; • Do not add the part reference for the root node when adding associativity to stroked PMI. • Do not add associativity for stroked PMI when doing extended PMI or else the associativity gets duplicated.
CATIA V5-6 to JT	For facetted solids with a large number of triangles, split into multiple JtkShapes to prevent a JT export issue with large shapes. Previously the exported JT part was empty when shapes with a large number of triangles were created (> 19 million). On advice from Siemens the maximum number of triangles is limited to 500,000 but this can be changed using advanced command line option "**max_shape_size <value>**"
CATIA V5-6 to JT	Correction to the creation of Coordinate Systems. The input to the JT function to create the coordinate system (CSYS) implied that the top-left and bottom-right coordinates had to be supplied, whereas it was actually the x and y vectors.
CATIA V5-6 to JT	Correction to the positioning of assembly components when creating a MONOLITHIC assembly with SUBNODE sub-assemblies.
CATIA V5-6 to JT	Make sure that the "enhance colour" option is only applied to faces. Previously, it was being used for all Materials and a problem was showing up for facetted solids.
CATIA V5-6 to JT	The attribute "JT_PROP_MEASUREMENT_UNITS" is now added to every node instead of being added to JT Parts if mass properties have

	been requested. It is set to a character string defining the output units (e.g. "millimeters").
CATIA V5-6 to JT	Fixed issue with Coordinate System processing, where axis vectors were incorrectly returned.
CATIA V5-6 to JT	Allow user control of exporting V5 hidden geometry
CATIA V5-6 to JT	Fix for 3D annotations read from some 3DXML files.
CATIA V5-6 to JT	Redesigned Tools Options page for JT Export to incorporate new spec_tree option. This option groups bodies as defined in top level CATIA specification tree names. Requires DS Hotfix for Service Pack 4 of V5-6R2013 & V5-6R2014
CATIA V5-6 to JT	Added support for "**use_instance_names**" where the JT node names are from the GCO structure occurrence names as opposed to the structure detail names only.
CATIA V5-6 to JT	Added support for "**name_root_node**" where the JT root node name is from the GCO DESIGN entity as opposed to the output file name. (N.B. for PER_PART, FULL_SHATTER and MONOLITHIC structure types, using this option will cause the output JT file name to change.)
CATIA V5-6 to JT	Allow for the case of no assembly structure when creating PLMXML output. The PLMXML file now correctly references the JT Part.
Multi-CAD Architecture	Implemented new read/write architecture and removed XCAD JT Server process
Multi-CAD Batch	Added separate _BATCH file processing license to control batch processing usage
Multi-CAD Batch	Add simplified command line options -o <outfile> mode_visu mode_visu_snap mode_catpart Also updates to V5Part OutputDocument type options setting to align with JtToNav bug workaround. Options now set directly on the InputDocument in this case.
Multi-CAD Batch	Implement XCAD_BATCH_MODE and implement TSC_IGNORE_JT_CATSETTINGS for JT in batch mode -o usage for JT V5 import executable.
Multi-CAD Menu	New optional CATIA Menu item File > Theorem Command. Plus environment variable to control the process that is launched TS_CATIA_MENU_LAUNCH_COMMAND=%TS_INST%\bin\CADverter.cmd
Multi-CAD Menu	Enhance generation of progress files to include suitable XML format output that is used by the Theorem User Interface.

Known Limitations and Restrictions

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

Ref ID	Limitations and Restrictions
1	*Minimum Service Pack + Hotfix Level:* There are a number of issues related to the use of the Multi-CAD infrastructure with lower Service Pack levels of the CATIA V5 installation. For example for both CATIA V5-6R2013 & CATIA V5-6R2014, SP4 is the minimum level that DS recommend for optimal functionality. In addition DS have also introduced a number of improvements in their Multi-CAD architecture that require additional Hotfixes to be implemented over the SP4 installation. For the latest Hotfix information please contact your local DS support agent.
2	*Running the JT to V5 translator via the Theorem UI:* There is a known limitation imposed with this release which means that both the creation of VISU & VISUSNAP output format data is not supported when using the Theorem UI. This restriction is NOT imposed when either running the translator interactively within V5 or when the process is run from the command line. Therefore please limit the use of the Theorem UI for JT to V5 when creating geometry in CATPart format.

Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID#	Product Impacted	Description
CAS-02146	CATIA V5-6 to JT	Previously the property "CADSource" was added to all nodes via a separate piece of code where a node was created. These pieces of code have been removed and the new property "CAD_SOURCE" (ISO standard name) is now added in a single location. This change also allows the new property to be mapped, in the same way as all other properties.
CAS-02180	JT to CATIA V5-6	Fixed issue with the rootInstanceRef parameter on the InstanceGraph element in addition to the rootRefs parameter. e.g. <InstanceGraph id="root" rootInstanceRef="IDrootInst0">
CAS-02254	CATIA V5-6 to JT	Encode all GCO text attributes as UTF8 when writing to PLMXML. Previously, unusual characters (e.g. e acute) were not encoded correctly. If the attribute was used as a file name (e.g. MPARTNAME) this caused the reference in the PLMXML file to be incorrect.
CAS-02278	JT to CATIA V5-6	Allow tessellation parameters to be supplied to tessellate ULP data. Two new command line arguments have been added:- tess_ulp : Stops the automatic tessellation and invokes configurable tessellation. -z "config_file" : To supply a JT config file containing the tessellation parameters. Only the LOD 0 settings are used.
CAS-02376	JT to CATIA V5-6	1. Addition of "ts_cfile" command line option. 2. Addition of TS_ATTR_VERSION property to DESIGN entity 3. Rename non-unique details from PLMXML files.
CAS-02378	JT to CATIA V5-6	Allow for "Header" elements at the top level that do not contain any structure - in this case get the structure from the "ProductDef" element.
CAS-02395	JT to CATIA V5-6	Attempt to get the colour right for tessellated solid read. The Part, Shapes and TriStripSets can all be coloured and it's difficult to determine what colour VisMockup will display

CAS-02436	JT to CATIA V5-6	Correct error when modifying the names of details to be unique. When comparing a detail name with the existing names a match was found with any part of the name when it should have been checking from the beginning only.
CAS-02483	JT to CATIA V5-6	Added filtering of PMI via PMI types supplied in a file. New command line: **pmi_filter_file *<filter_filename>***
CAS-02502	JT to CATIA V5-6	Fixed an issue when reading a rotation matrix for an Occurrence transform, the matrix needs transposing before storing in GCO.
CAS-02557	CATIA V5-6 to JT	Make XT the default type for BREP geometry. A new command line argument, "**jt_brep**" has been added to enable the previous representation to be selected.
CAS-02564	CATIA V5-6 to JT	Change estimate of the number of Parasolid tags to be created when processing open solids to use the no of faces in the solid. The previous method underestimated when used in the V6 translator and caused a crash.
CAS-02584	CATIA V5-6 to JT	Added **AdvancedOptions, UseInstanceNames and NameRootNode** to the options that can be specified in a config file. **AdvancedOptions** is a special type that allows a string of space delimited advanced options to be input. It is not possible to supply file names containing spaces via this method.
CAS-02604	CATIA V5-6 to JT	Correction to prevent a crash if a DIMENSION2 text block has a NULL frame and PMI properties are being created (extended_pmi)
CAS-02606	CATIA V5-6 to JT	When processing model-based definition, mass properties are now processed directly as supplied in the GCO data. Previously for JT Parts that were created from a group object, the mass properties were accessed from the node of the original CATPart. This has been done because the V5 read leg now applies the properties to the relevant group objects.
CAS-02620	JT to CATIA V5-6	Added the ability to sanitize node names, specifically for use with the Theorem User Interface to remove '/' characters that are interpreted as folder delimiters. Two new command line arguments "**sanitize_chars *[chars]***" and "**replace_char *[c]***" have been added

CAS-02633	CATIA V5-6 to JT	Added line style and line width to both precise and tessellated wireframe
CAS-01907	CATIA V5-6 to JT	Change license features from Catia5_xxxx to XCAD_xxxx for all XCAD V5 export products. The license code letter 'y' denoting Catia5 changes to 'l' denoting XCAD.
CAS-02143	JT to CATIA V5-6	Migrate CATIA V5 CAA fixup routines into base library to allow usage in XCAD servers.
CAS-02143	CATIA V5-6 to JT	On reading CATIA bodies, add checks on gco brep references for invalid ids using check_brep_ent_range() Remove problem faces using remove_face_from_brep()
CAS-02173	CATIA V5-6 to JT	Added fix for surface read error which was not cleaning bad face from conversion. Added fix for bad face edge list data with repeated edge sequence and incorrect edge data.
CAS-02240	CATIA V5-6 to JT	Fix circle read to check arc start and end points in check for existing gco conic. Failed to create multiple arc segments from single CATIA circle with different trim limits. Fix handling of 'convert_curves' option in xcad_jt_opts.txt.
CAS-02241	CATIA V5-6 to JT	Fix circle read to check arc start and end points in check for existing gco conic. Failed to create multiple arc segments from single CATIA circle with different trim limits. Fix handling of 'convert_curves' option in xcad_jt_opts.txt.
CAS-02328	CATIA V5-6 to JT	Filter out insignificant error messages when analysing XCAD err log file for summary and error code data in conclusion of processing.
CAS-02328	CATIA V5-6 to JT	Fixed View associated annotations support. Incorrect annotation id update value was pointing to dim container DITTO not DIMENSION2 entity. Code in place to handle this had a bug. For 3DXML input, associative dims are handled in child structure, not direct associations as per CATIA.
CAS-02362	CATIA V5-6 to JT	Correct export log file process to use the correct vwr_xx log file name which is based on the viewer_<output_part_name>.log form. Previously incorrectly using viewer_xx.log where xx was JT NX, PROE.
CAS-02401	CATIA V5-6 to JT	Add capacity to read Alpha (opacity) value when available in the interface CATIExchangeExtendedGraphicProperty. SR00304278 - Availability is planned for V5-6R2014 SP7 , V5-6R2015 SP04 HF49, V5-6R2016GA.
CAS-02401	CATIA V5-6 to JT	Add new 3DXML write option write_tess, to handle the output of geometry where no BREP data has been read. XCAD_GEOMETRY_TYPE is set to XCAD_TESSELLATED.
CAS-02401	CATIA V5-6 to JT	Fix to prevent crashing if TS_INST folder contains space characters. CATIA V5-6 to JT config file open failed.
CAS-02422	JT to CATIA V5-6	Fixfor crash while importing jt models with PMI.

CAS-02434	CATIA V5-6 to JT	Add support for mask file read for layer masking. Entity masking is currently supported via separate options.
CAS-02500	JT to CATIA V5-6	Redesign XCAD Importer to replace use of xcad server reader process with managed invocation of XXX_vwr cmd process followd by gco file import and then TSCCADWriter input geometry.
CAS-02525	CATIA V5-6 to JT	Multiple facet solids (FSOLID) read from CGR files are incorrectly positioned. A bug in managing nested orientation info was fixed. Also bug in translation calculation fixed.