PRODUCT RELEASE ANNOUNCEMENT

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

Document Type	Product Release Announcement
Status	Released
Revision	3.0
Author	Product Manager
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Contents

History

Revision	Update Information
1.0	Initial Release
2.0	Updated list of resolved support calls
3.0	Updated for Daimler Compliance

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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-S5	CATIA V5 (CAA Based) to JT (uni-directional) with PMI Add On
CA5JT-S6	CATIA V5 (CAA Based) to JT (uni-directional) with Electrical Add On
CA5JT-A7	CATIA V5 (CAA Based) to JT (bi-directional) with 3D PDF Add On
CA5JT-A8	CATIA V5 (CAA Based) – JT add-on STEP AP242 BOM assembly module

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 19.3 CATIA V5 (CAA) - JT CD image
Theorem Unified Interface installation media	Download Version 19.3 Unified Interface CD image

New Features / Enhancements

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

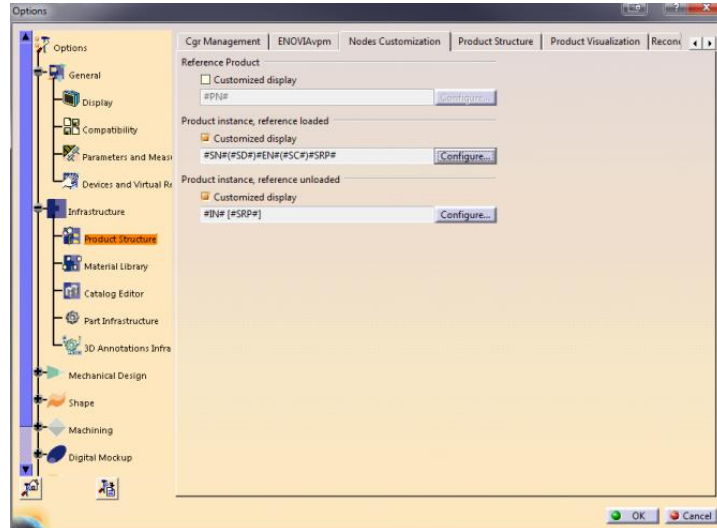| Product Impacted	New Feature / Enhancement Description
JT to CATIA V5-6	Added an advanced command line option to remove all JT properties from all nodes in the JT file (**strip_props**)
JT to CATIA V5-6	Additional JT utilities provided to analyse the content of a JT file. These are **jt_content.cmd, 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	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	Tessellation from existing LOD data can optionally be stored in the FSOL2 entity for the STEP write leg. The following command line arguments have been added; • **tess_brep**: To produce tessellation directly from the BREP data • **tess_and_read_brep** : As tess_brep, with the BREP data also stored with additional links between the BREP faces and the faces in the FSOL2. • **fsol2**: Store tessellation as a GCO FSOL2 entities.

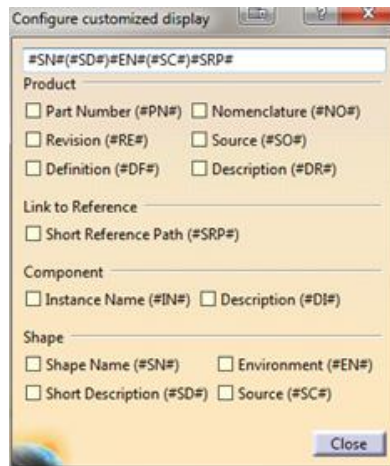|	
JT to CATIA V5-6	Change to optionally store PMI strokes as pcurves to allow for non-planar PMI. Advanced command line option: **pmi_pcurves**.
JT to CATIA V5-6	Change to prevent negative PMI ids from being used as GCO HEID's, causing the write leg to ignore.
JT to CATIA V5-6	Cope with the situation 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.
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	Write Opacity to CGR
JT to CATIA V5-6	Added creation of CAT3DAxisRep in CGR
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	Extend ignore_failed_faces & process_failed_faces to include write_failed_faces ignore_failed_faces - Doesn't write faces for failed BREP or Open solids write_failed_faces - Creates faces for failed BREP or Open solids but doesn't try to fix up process_failed_faces - Trys to fixup and create faces for failed BREP or Open solids
JT to CATIA V5-6	HIDE_FAILED_GEOMTRY Hides bodies that are thought to have failed
JT to CATIA V5-6	SHOW_FAILED_GEOMTRY Shows bodies that are thought to have failed
JT to CATIA V5-6	body_per_part / part_per_solid Creates a separate CATPart per body
JT to CATIA V5-6	ignore_mfcl_checks Turns off checks on making 2D face curve list
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.

CATIA V5-6 to JT	INSTANCE_RENAMING V5 This causes the JT output tag names to be altered in accordance with the Tools->Options->Infrastructure->Product Structure->Nodes Customerization panel The possible items include The code looks for the #XX# type string finds the appropriate attribute on either the instance or associated detail and replaces it in the string. Caution! When using this option you potentially will create an assembly with both duplicated item names and/or empty item names. This option is only supported when running the translator from the command line.

CATIA V5-6 to JT	INSTANCE_RENAMING <STRING> where <string> can be combination of :-
	- Product,
	#PN# - Part Number,
	#NO# - Nomenclature,
	#RE# - Revision,
	#SO# - Source,
	#DF# - Definition,
	#DR# - "Description",
	- Link to Reference,
	#SRP# - Short Reference Path,
	- Component,
	#IN# - Instance Name,
	#DI# - Description,
	- Shape,
	#SN# - Shape Name,
	#EN# - Enviroment,
	#SD# - Short Description,
	#SC# - Source
	Setting the option to V5 will replicate the naming style from the CATIA V5 CATSettings
CATIA V5-6 to JT	REPRESENTATIONS Causes all representations for a node to be read
CATIA V5-6 to JT	REPRESENTATION <name> Causes the names representation for a node to be read else the default representation if the node doesn't have the named one
CATIA V5-6 to JT	OUTPUT_MBD_LEAF_NODE Reads extra level of specification tree structure
CATIA V5-6 to JT	READ_PART_PMI Reads just part PMI

CATIA V5-6 to JT	READ_PART_ASSY_PMI Reads part pmi that's in an Assembly set
CATIA V5-6 to JT	READ_ASSY_PMI Reads assembly & part pmi that's in an Assembly set
CATIA V5-6 to JT	READ_GEOMETRY_EDGES Causes the edge curves for a planar FTA construction geometry to be created as standalone curves allowing the plane to be views edge on.
CATIA V5-6 to JT	SOLID_COLOUR as option to disable reading of face_colours
CATIA V5-6 to JT	WASH_COLOUR Washes colour specified on the root node of the specification tree down over bodies
CATIA V5-6 to JT	CREATE_THREAD creates the wireframe geometry associated to a FTA Thread construction geometry
CATIA V5-6 to JT	QUASI_CYCLIC_TOL <tol> To control the tolerance that a surface is checked for quadi cyclicity
CATIA V5-6 to JT	DATUM_FACES <size> Planar datum geometry is displayed in V5 as a parallelogram sized according to the screen (i.e. doesn't change as you zoom in or out) But the supporting geometry is infinite sized according to the model size but this can overwhelm systems that use the bounding box for fitting the geometry to the screen. This option allos this geometry to be read as a face and sized to a user defined size.
CATIA V5-6 to JT	MERGE_CGR_VERTICES Stopped the merging of CGR vertices this option reenables
CATIA V5-6 to JT	PMI Wrapper now installed by default
CATIA V5-6 to JT	Fixed issue with reading mask files created in new UI (seen with solution to CAS-02331)
CATIA V5-6 to JT	Honoured transparency inheritance from product node in CATPart (CAS-02337)
CATIA V5-6 to JT	V5 PMI Read now notes positional and orientation links
CATIA V5-6 to JT	V5 PMI Read now notes PMI – PMI association for GDT's, Simple Datum's & Datum Target's
CATIA V5-6 to JT	Added units to attributes
CATIA V5-6 to JT	Get attribute magnitude from attribute mapping file
CATIA V5-6 to JT	Store part root colour and opacity on detail

CATIA V5-6 to JT	Add axis systems to groups when OUTPUT_MBD on
CATIA V5-6 to JT	Update radius if FTA scaled and start and end angle if rotated
CATIA V5-6 to JT	Added dedicated OUTPUT_MBD – Now reflects OUTPUT_MBD state in options list rather than just the subordinate options GROUPS_ON & PARAMETERS
CATIA V5-6 to JT	Allow opacity on solids / faces to be inheritted from root node of specification tree
CATIA V5-6 to JT	Improvements in reading multiline text in NOA in regard to positioning and justification
CATIA V5-6 to JT	When reading a CATProduct that references a CGR that's not on the system as from DELMIA save the CGR file to a temp location, read then delete
CATIA V5-6 to JT	Sets the IDENTICAL_POINT_TOL based on the CATIA V5 model size
CATIA V5-6 to JT	For Datum Targets remove any duplicated geometry assoication when also associated to Simple Datum
CATIA V5-6 to JT	Improvements in reading FTA Construction geometry when referenced from a CAPTURE and OUTPUT_MBD is active
CATIA V5-6 to JT	Improvements in honouring geoemetry hide / show state from a CAPTURE if OUTPUT_MBD is active
CATIA V5-6 to JT	Updated to include CAD_MASS_UNITS in each node to meet Daimler compliance

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	*CATIA V5 Import/Export Menu:* The configuration files required for the current Import/Export menus are compatible with the old style CADverter UI configuration files and not those created by the latest Unified Interface.
3	*Support for standalone CATIA V4 processing.* There are no specific CATIA V4 to JT translators included on the media with this release. However the CATIA V5 to JT translators are capable of processing standalone CATIA V4 .model files with an appropriate Theorem license.

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Resolved Support Calls

The following list of Support Calls are resolved with this release

Support Call ID	Brief Description
CAS-01491	Overlapping GDT Above Text
CAS-01797	ETS Data mismatch Between Catia and vismockup
CAS-02057	Failure when processing CATIA V5 Scanned data to JT
CAS-02071	17.3 V5 to JT Fails reading PMI and other PMI issues
CAS-02110	ETS JT showing surface issues for Catia part
CAS-02117	ETS JT showing surface issues for Catia part
CAS-02120	CatiatoJT failing with different Exit code
CAS-02121	Catiatojt failing with different Exit code
CAS-02140	CATPrt to JT creates a PER_Part structure and is not properly being viewed
CAS-02142	JT file does not match the V5 data
CAS-02146	Faces missing in 17.3.006, possible regression
CAS-02159	We ran using convert curves option and earlier version of theorem. Still we gear tooth was not smooth as in CATIA data.
CAS-02188	JT file with multiple bodies
CAS-02198	PMI Issues - resultant jt is not showing arrows for the PMI datas (Text, flagnote, geometrical tolerance).
CAS-02202	We could not able to create for JT for the item JX6A-18E256-A/4.
CAS-02203	Could not able to create for JT for the item DC-U55X-jl14-040100-18001/6
CAS-02246	JT file has missing faces
CAS-02261	JT file missing symmetrical data
CAS-02266	Catia5-JT-3DPDF pmi issues
CAS-02268	Incorrect JT assembly generation from V5 data -
CAS-02288	JT files are not getting created for the specified items
CAS-02290	Translation takes 4 hours in standalone and production it takes more than 5 plus hours.
CAS-02321	CATIA V4 to JT - Translation issue - Geometry output does not match input
CAS-02322	CATIA V5 /V4 to JT - Translation issue - Geometry does not match
CAS-02323	CATIA V4 to JT - Translation issue - Geometry output does not match input
CAS-02324	CATIA V5 to JT - Translation issue - Translator Crash
CAS-02325	ETS translation failed for item: DC-CD42-040100-01000/12.
CAS-02331	Overlapping text output from multiline FTA text
CAS-02334	Issue converting from V5 to JT
CAS-02337	Transparent JT file
CAS-02352	When an annotation is hidden in CATIA, the same annotation should be hidden in TC VIS.
CAS-02354	JT goes to Terminal on submitting a manual request and JT is not created in the staging as well
CAS-02369	MPROPS - Variance
CAS-02385	ETS Catia to JT Issue-translation went to terminal state for a particular assy data

CAS-02407	Item ID: HP53-8472-A translation fails with crashing
CAS-02430	Unable to see contents of JT file in TcVis
CAS-02431	Unable to open created JT file in TcVis
CAS-02436	The following part has empty JT for the CAD data.
CAS-02437	JT file not created
CAS-02438	JT created for the item W719174-S, but there are some extra surfaces visible
CAS-02445	When JT file H1RO-7E395-A/8 is opened in Vismockup an error pop-up is displayed
CAS-02476	The original JT only displayed a line
CAS-02487	Regression report by Ford AD
CAS-02489	Failure to process PLMXML to V5 using existing CGR files
CAS-02503	V5 to JT text orientation appears to be flipped
CAS-02510	For the part, in VISMOCKUP only a line is visible.
CAS-02515	CATIA and JT files do not match.
CAS-02519	Failure JT Tess to CGR
CAS-02521	Catia to JT translation fails with a catastrophic error.
CAS-02530	Failure to process assembly and colour
CAS-02543	Publish 3D fillets incorrect position
CAS-02545	Rugosity is inverted in some cases
CAS-02552	V5>CreoView: unwanted lines appearing in CreoView output
CAS-02564	Problem with an Assembly-Structure converted to PLMXML-Structure
CAS-02587	For JTs created with the new version of theorem (18.2.124) the color of the lines are set to black
CAS-02594	JT creation is failing in all versions of Theorem
CAS-02603	CAD data is producing several surface errors and has failed faces
CAS-02604	Solid fsilure when writing data in XT_BREP format
CAS-02617	JT not created, Possible regression issue in 18.2.126
CAS-02620	JT not matching CATIA Linestyles and colors
CAS-02623	JT to V5 Slow performance opening only tessellated JT's