

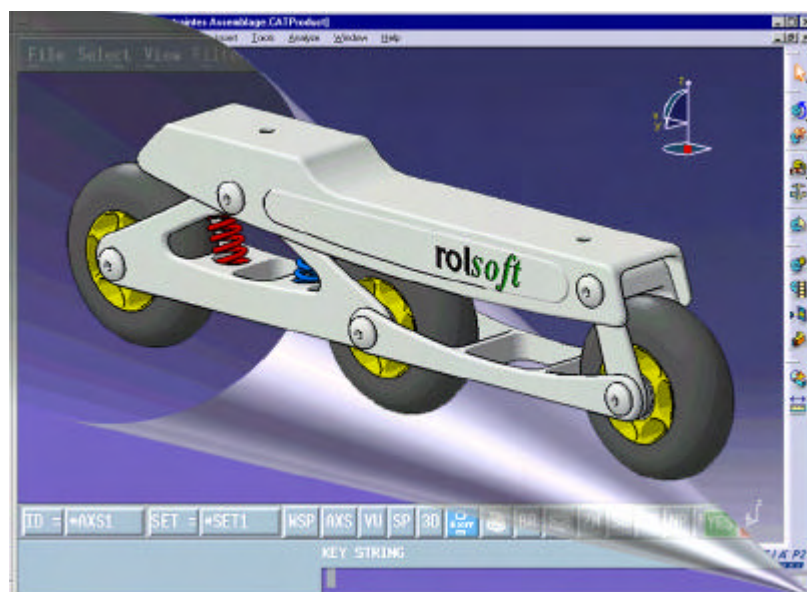


# CATIA V4/V5 Interoperability

White Paper – Version 5 Release 11

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## 1 Introduction

This document is intended for customers, Business Partners and IBM specialists who have to deploy CATIA V5. To fully comprehend this document the specialist requires knowledge of the CATIA V4 product and a global understanding of what “V4/V5 Integration” represents. Its purpose is to clearly expose the extent of V4/V5 Integration in the context of V5R11 and explain, when available, what will be proposed in the future. These plans are subject to change.

## 2 V4/V5 Integration Definition

How can CATIA V4 Data and Applications operate in a CATIA V5 environment and vice versa?

Integration covers three aspects:

- data interoperability (V4 data usage in V5 and vice versa)
- application interoperability (how can V4 and V5 applications work together)
- platform interoperability (UNIX and Windows scenarios including V4/V5)

Integration capabilities allow customers to use data without migrating it.

## 3 V4/V5 Integration Highlights

The best approach for handling interoperability is to use the product CATIA - V4 INTEGRATION 2 (V4I), through the "CATIA Site Navigator", which allows the seamless display of CATIA Version 4 Data from the Version 5 environment, regardless of the platform used, Windows or UNIX.

This viewer allows users to:

- display the CATIA V4 model with both a graphical representation and a structure tree
- selectively copy CATIA Version 4 data and paste it into a new part definition
- navigate across CATIA V4 model workspaces or geometric sets, thus providing a powerful tool to review and understand the provided V4 data.
- have full zooming, panning and rotation capabilities.
- inherit all of the advanced shaded or non-shaded viewing options of Version 5
- display data using the viewpoints stored within a CATIA V4 model
- manage visual properties such as show/hide, colors
- layer/filter configuration
- access V4 workspaces
- print the data representation
- preventively check if CATIA Version 4 elements are available in V5 before a copy-paste operation (Check-Icon in V4I Navigator)

CATIA Version 5 offers an upwardly compatible environment allowing the **re-use of native V4 data** within the following CATIA V5 functionality:

- Create a CATIA V5 generative drawing from a native CATIA V4 model (supports Solids and Skins; Faces and Surfaces only supported in CGR mode)

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- Direct reading and automatic transfer of native V4 Session documents into CATIA V5 CATProduct structures.
  - Direct reading and automatic transfer of native V4 assembly structures into CATIA V5 CATProduct structures.
  - Use combinations of CATIA V5 and native CATIA V4 data within a CATIA V5 CATProduct definition.
  - Copy-paste native CATIA V4 3D data into a new CATIA V5 CATPart definition, thereby completing a CATIA V5 design
  - Copy paste native CATIA V4 2D data into a new CATIA V5 CATDrawing definition, thereby completing a CATIA V5 design
  - Migration of models can be done either interactively by copy/paste or in batch
  - Transfer of CATIA V4 3D Library into CATIA V5 Catalog in batch
  - Transfer of CATIA V4 Library of Draw Details into CATIA V5 Catalog of CATDrawings in batch mode

In addition, some **downwardly compatible capabilities** are provided to allow the use of CATIA V5 data in CATIA V4 (Release CATIA V4.2.0 Ref. 1 or higher) as follows:

- Read, and use for reference, a CATIA V5 CATPart as an exact solid in passive mode in CATIA V4
  - On a CATIA V5 CATPart loaded in a CATIA V4 session as a passive model, the following functionality can be used:
    - V4 Generative Drafting (AUXVIEW2 function).
    - V4 Analysis.
    - V4 Meshing.
    - The NC functionality on V5 CATParts is provided since V5R4 and V4R23.1
- Define a V4 SolidE Import Primitive referencing a CATIA V5 CATPart body and keep V5 associativity in V4 using SolidE/LINKS capability.
- Convert a CATIA V5 CATPart to a V4 exact SOLID model for use in different V4 functions (BREP only).
- Convert a CATIA V5 CATProduct to a V4 Session.
- Data migration can be done either interactively by using “Save as” with type \*.model or \*.session, or in batch.

In addition to the V4I product , the CATIA V4 INTEGRATION 1 (V41) product provides V4 integration capabilities for P1 configurations. It allows reading and display of V4 data, migration of V4 2D data to V5, and exploding of V4 sequential files (.exp, .dlv3) into V4 models via batch job. Contrary to the V4I product, the V41 product does not support migration of V4 3D data or saving V5 data in V4 data.

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## **4 General Environment Data**

### **4.1. Project File: Reading / Transferring / Using a V4 PROJECT file**

- There is no PROJECT file in V5 as defined in V4. The company standards and rules will be managed through the use of «styles» in «standards» in the area of drafting.
- It is possible to read a V4 drawing referencing an internal or an external PROJECT file on both UNIX and Windows.
- An External PROJECT file is readable directly under Windows (see Windows versus UNIX Limitations at the end of the document); it can also be referenced using a DLNAME.
- As of today, no information can be transferred from the PROJECT file to V5 (company standards migration scenario).
- The contents of the PROJECT table cannot be browsed directly with the CATIA Site Navigator (CATIA V5 browser for V4 Data). However, visualization of supported project table entries will be displayed with the Site Navigator when browsing models that reference project table data.
- V4 user-defined fonts are supported for visualization of V4 models in V5 except for G&D texts.
- V4 user-defined line fonts can be browsed and migrated into V5.
- V4 cell patterns can be browsed and migrated to V5. In V5R11 a variable has to be set to support the migration (V4MigrCellPattern=1).

### **4.2. Converting V4 documents with names containing V5 forbidden character**

It is possible to specify a character-mapping table and to reference it in V4 Integration settings in Tools -> Options -> General -> Compatibility. This table will be considered each time a V5 document is created from V4 data in the following functionalities:

- Batch conversion.
- File/open .session
- File/open .asm

CATIA V5 only accepts standard ISO646 characters in document names. Other characters will be replaced automatically, possibly with the replacing character specified in the user table. If no user table is specified, a default table is used.

The default table should be consistent for special Windows forbidden characters. If model names contain special national characters it is recommended to create a dedicated table.

NOTE: this map is used for naming generated V5 documents. OS restrictions regarding the native opening of files named with illegal characters will not be solved. For instance a model named

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“ILLEGAL: NAME.model” will still not be readable on Windows, but it will be readable on UNIX and the batch conversion on UNIX will generate a document named “ILLEGAL\_NAME.CATPart” (by default, or any replacement character specified in the user table), which is V5 compliant.

### **4.3. Reading V4 documents with names containing V5 forbidden characters**

Limitations still exists on Windows when opening specific V4 documents:

- \*.session, containing a reference to a document which name contains V5 forbidden characters
- \*.model, which name contains V5 forbidden characters.

Using a batch program, a V4 document and its dependencies can be renamed into a file, which can be read on Windows. The batch program takes into account the Windows character-mapping table to rename the documents. Note that the documents are only renamed and not modified. The links contained in the documents will be automatically converted into a V5 session with a Windows compatible name.

The goal of this batch program is to allow to open in Windows a session that references \*.models with forbidden characters and to keep the links between such documents, or to open a model which name is not Windows compatible.

The batch program allows the managing of the level of dependencies between the documents.

The links taken into account are:

- Session links
- SolidE import primitive links

Due to the forbidden characters, it must be launched in UNIX.

Usage: CATV4ToV5NTCompatibilityName *-arg*

### **4.4. Models: Reading / Transferring / Using V4 Models**

It is possible to read a V4 model in a “light mode” (ignoring the Draw data and loading the 3D data only). Light mode can be activated through Tools -> Options -> General -> Compatibility Settings. Note that this mode can lead to a conflict during a V4 to V5 migration of draw data. Light mode is taken into account when sending V4 documents to CATIA V5 through PDM applications (i.e. LCA, VPM).

### **4.5. Libraries: Reading / Transferring / Using V4 Libraries**

- It is possible to read a V4 model referencing a V4 library.

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- In V5, the users can navigate through 3D V4 libraries using the CATIA Site Navigator and copy/paste the geometry of solids of a detail to a part.
  - CATIA V5 introduced the concept of CATALOG to support standard parts (i.e. the classification and access of V5 objects such as CATParts, CATProducts, CATDrawings,...).
  - A batch migration tool is provided for families of space and draw details. The families and keywords are converted into V5 Catalog classifications.
  - Space details are converted as V5 Catalog Parts.
  - Draw details are converted as V5 CATDrawing Catalog.
  - Symbol Libraries are not converted in V5R11
    - Symbols in V4 2D drawings are converted into V5 details during the migration of the model and can be stored afterwards in V5 Catalogs.

#### **4.6. Plot sheet data: Reading / Transferring / Using a V4 Plot Sheet**

- Visualization and print capabilities for V4 plot sheet data are currently not in plan.
- Transfer capability for V4 plot sheets is not planned.

#### **4.7. Sessions: Reading / Transferring / Using a V4 Session**

- V4 Sessions can be imported into a V5 CATProduct Structure file. However, models contained in the V4 Session saved with the "save all data" or "save modified data" option, will not be referenced in the resulting V5 Product Structure.
- It is possible to open a V4 Session as "reference only". It will be converted automatically into a V5 CATProduct file. Positioning of the models is preserved. Sessions using DLNAMES are supported.
- It is possible to migrate V4 Session as V5 CATProduct in batch mode (Sessions which are saved as 'reference only').
- It is possible to save a V5 CATProduct as V4 Session. Referenced V4 models of a V4 Session are also migrated into V5 CATParts.

#### **4.8. Assemblies: Reading / Transferring / Using V4 Assemblies**

- V4 Assemblies (.asm) can be read and are converted automatically to a V5 CATProduct Structure file.

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- Positioning and constraint-related information will be preserved.

#### **4.9. Attributes: Reading / Transferring / Using V4 Attributes**

Attributes are mainly used for CAA applications. There is no obvious mapping of V4 attributes to V5 because V5 CAA applications should be re-designed. It is planned for R12 to print attributes into the migration report where they can be extracted through CAA V5 applications.

#### **4.10. Thickness: Transferring V4 Drafting Data**

There is a mapping between V4 thickness to the nearest V5 thickness. To be sure to get same thickness for V5 element, initialize the V4 Standard Thickness:

- For browsing: Definitions have to be done in :  
Tools -> Options -> General ->Display -> Line Thickness
- For migration: Definitions have to be done in:  
Tools -> Standards

#### **4.11. Linetype: Transferring V4 Drafting Data**

There is a mapping between V4 linetype to the same V5 linetype. V5 linetypes have to be defined corresponding to the V4 UNSPEC linetypes:

- For browsing: Definitions have to be done in :  
Tools -> Options -> General -> Display -> Linetype
- For migration: Definitions have to be done in:  
Tools -> Standards

#### **4.12. Model Comment: Reading / Transferring / Using V4 Model Comment**

- Model comment of 3D Models can be browsed as a property of the Master entity. This can be done from the CATIA Site Navigator or from any Product centered workbench.
- V4 3D model comment is migrated as the Product Description Property of the resulting V5 document. Detail comments are also migrated as Product Description properties. The migration is available in batch mode and interactively with copy/paste (by selecting \*MASTER).

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## 5 Visualization of V4 Models in V5

### 5.1. Drafting Elements

- Layers and filters are preserved but cannot be modified.
- DRO data (V3 Drafting created in V4) can be visualized in V5.
- When browsing a 2D model, it is possible to add several additional models to simulate the V4 overlay functionality (File -> Open with “Drafting On”). The result can also be printed.
- A light mode offers to open a \*.model without loading and visualizing drafting elements.

#### 5.1.1. Raster Data

V4 models containing raster data can be browsed through the CATIA Site Navigator.

### 5.2. 3D Wireframe / Surface / Logical Elements

With some exceptions all of the above entity types may be visualized in V5. Nets and SPACE constraints (\*CST) cannot be visualized.

### 5.3. Solids

SolidM and SolidE can be displayed in V5. All visualization modes (HLR,SHD, ...) are supported.

### 5.4. Application Elements

- Application-specific information, e.g. mesh elements, kinematics elements, and covering elements cannot be visualized.
- 3D geometric data related to Engineering applications such as tubing, piping, schematics, structures, electricity 3D and AEC primitives can be visualized (see Downstream Application Data paragraph).

### 5.5. User-Defined Elements

Visualization of user-defined elements (UAE's, UGE's) is available.

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## 5.6. Pick/No Pick Properties

Pick/no pick properties are not kept. This capability is in the plan with a low priority.

## 5.7. Other Information

- Functional Dimensioning and Tolerancing information can be visualized in the V5 DMU product “Dimensioning and Tolerancing Review” only.
  - Parametric 3D information cannot be visualized. At this time, there is no plan to include this capability.
- File/Open a V4 \*.model containing 2D and 3D data in “Light Mode” ignores the 2D data and loads only 3D data.

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## 6 Transfer of V4 Models to V5 and Editing in V5

### 6.1. Considering V4 3D Layer/Filter

In an interactive copy/paste operation, a user can migrate the entire geometry to V5 or select only geometry laying on a specific V4 layer/filter. With V5R11 only 3D data is supported by that functionality.

### 6.2. Transfer of Drawings to V5

- Before starting the transfer of V4 2D drawings, the following standards have to be synchronized:
  - V4 DRWSTD Function Settings with V5 Tools -> Standard to receive the same dimension behavior
  - V4 GRAHIC/THICKNESS -> GRAPHIC/WIDTH with V5 Tools -> Standards
  - V4 GRAPHIC -> LINETYPE with V5 Tools -> Standards
- Pure Interactive V4 drawings can be transferred by selecting \*DRAFT or view by view to V5 CATDrawing documents through the copy/paste operation.
- Transfer of DRO data (V3 drawings created in V4) has to be done in 2 steps in V5R11: conversion from V3 to V4 Drawing format first (using /ANND34 or CATMOD in V4) and than transfer to V5.
- Geometry of Transparent Views are converted into 2D geometry in a V5 view.
- V4 reframed views are transferred as V5 reframed views.
- V4 raster elements are converted into V5 raster features.
- ISOCOMB tolerance formats are migrated into V5.
- V4 Dittos used with layer filtering are migrated to V5 multiple details structures corresponding to the layers filter; instantiation is done on the detail which is visualized in V4.
- In a single V4 model, DRAW data can reference several standards (ANSI, ISO,...). During conversion the data is transferred in the standard of the receiving V5 drawing.
- V4 Draw Detail/Dittos are transferred as V5 Draw Detail/Dittos.
- V4 Draw Symbol/Symbolo is transferred as V5 Draw Detail/Dittos.

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- Details and Symbols can be exploded when transferred depending on a setting.
  - Cell Patterns are migrated to V5. To activate that functionality in R11, a variable has to be set (V4MigrCellPattern=1).
  - User defined line fonts are migrated to V5.
  - Dimensions on 2D geometry are migrated as V5 dimensions except the dimension type is not supported:
    - Length on circle and curve, diameter on curve, diameter on cone.

This restriction means that such dimensions will be migrated as entity type “Graphical Dimension”.

- Non-associated V4 dimensions (isolated dimensions) are migrated into isolated dimensions in V5.
- In case of migration of V4 Interactive Drafting are two modes for dimensions are supported:
  - Semantics:
    - Properties and position are fully editable and associative to 2D geometry
    - Graphical behavior is taken from V5 Tools -> Standard
  - Graphical Dimension:
    - Not editable but with same representation as in V4
    - No associativity between dimension and geometry.
- In the case of migration of AUXVIEW generated 2D geometry:
  - Geometry is migrated as result, no 2D/3D associativity
  - Dimensions are supported in two modes:
    - Semantics mode:
      - Properties and position are fully editable but not associative to 2D geometry
      - Graphical behavior is taken from V5 Tools -> Standard
    - Graphical Dimension:
      - Not editable but with same representation as in V4
      - No associativity between dimension and geometry
- In case of migration of 2D geometry generated with AUXVIEW2:
  - Dimensions are supported in two modes (controlled by settings):
    - Semantics mode:
      - Properties and position are fully editable but not associative to 2D geometry
      - AUXVIEW2 dimensions can be manually rerouted to V5 3D geometry to create 3D/2D associativity. Isolated and no-isolated dimensions are visualized in different colors.
      - Graphical behavior is taken from V5 Tools -> Standard
    - Graphical Dimension:
      - Not editable but with same representation as in V4
      - No associativity to geometry
- Migration of Auxiliary Views is supported:
  - As\_RESULT:

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- Auxiliary views are migrated and the V4 geometry is transferred as Result.
  - No associativity between view and 3D geometry
  - As\_SPEC:
    - Auxiliary views are migrated and all types of Callout of Auxiliary Views are newly generated from V5 3D data.
    - Associativity between view and 3D geometry is kept and can be modified with Edit/Link capabilities.
    - Migration of Callout specifications from Section View Cuts and Detail Views takes into account all information defined in the V4 panel  
AUXVIEW2 -> CALLOUT -> GRAPH
  - Report generation (in CATReport directory) containing:
    - General information (Filename, mode, ...)
    - Geometry mapping (Element type in V4 to Object type in V5)
    - After migration, a view report is generated with following information:
      - View type
      - Specific operator (Breakout, Back clipping, Unspec Breakout)
      - Filter capability: UnCut, UnUse, UnHlr
      - Status: OK/KO/Not (Not means that the capability does not exist in V5)

### **6.3. Transfer of 3D Geometric Elements to V5**

- Axis of all types (fixed and unfixed) are transferred as Axis systems.
- Polylines and space constraints cannot be transferred. All of those are in the plan with low priority.
- Nets can be transferred through copy of \*MASTER or of the NET entity in the tree although they can not be visualized.
- Skin Design (\*SKD) elements are transferable as geometry.
- A first drop of GSM features are migrated as V5 GSD features.
- SURF2 elements are transferred as BREP only at the moment. The history transfer is planned in the future with a low priority.
- In V5, geometric elements are C2 continuous (curvature continuity). When transferring 3D geometry to V5 there is an automatic conversion from one V4 element to one or more C2 continuous V5 elements. It is possible to choose between optimizing the C2 continuity of the resulting V5 geometry and keeping the exact geometry (“Segmentation and Deformation” settings in Tools -> Options -> Compatibility -> V4V5 Space).
- Taking into account V4 element families optimizes conversion of space entities. This optimization leads to major improvements in terms of performances and result data size. The following elements are no longer converted:
  - Border curves of migrated faces

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- Surfaces support of migrated faces
  - Faces included in a migrated skin or volume
  - Volumes or skins used in solids.
  - By default, elements in No Show are not migrated except Exact and Mock-Up solids. They can be migrated while exporting the KEEP\_HIDDEN\_ELEMENT=1 variable.
- It is possible to activate a topology automatic cleaner (cleanV4Topology=1). The cleaner has following cleaning actions:
    - Degenerated edges or faces are eliminated
    - Partial degenerated faces are simplified
    - Some cases of “z” edges are cleaned by projection of the adjacent edge.

#### **6.4. Transfer of Mockup Solids to V5**

- Non-isolated Mockup solids:
  - Migration as RESULT creates CGR format which can be visualized when attached to a V5 CATProduct Structure though “FILE -> Add Existing Component”).
  - As\_SPEC migrates the SolidM into a V5 CATPart.
- Isolated Mockup solids:
  - Migration as RESULT creates CGR format.
  - As\_SPEC migrates the SolidM into a V5 CATPart when variable CATMigrSolidMUV4AsPart=1 is activated. That is supposed to be used for electrical elements only.

#### **6.5. Transfer of Exact Solids to V5**

- There are two options, in the Paste\_Special menu, to transfer a V4 exact solid to V5:
  - Transfer of the solid’s CSG information (As\_SPEC)
  - Transfer of the solid’s geometry only (As\_RESULT)The default option is “As\_RESULT”.
- It is possible to specify in V4 Integration SPACE settings whether the paste step and update step should be performed separately or in one unique transaction. The separation between the paste and update steps has been kept mainly to ensure the non-regression of migration scenarios. However the new integrated mode should be preferred for the following reasons:
  - It spares one user interaction.
  - The conversion report includes update information. This is not possible if the paste and update phases are separated.
  - Automatic reroute of broken contextual features (fillets etc.) is performed whenever possible. This is one of the main causes of errors during the update after pasting V4 specifications.

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- The conversion report includes a comparison between V4 and resulting V5 Breps/Features (volume, wet surface, gravity center). Activation through “REPORT\_BREP\_INFO=1”.
  - With V424R1 a V4 IUA macro /chktop allows a topology check to visualize potential problems, such as non-joined faces, thin faces or too sharp edges.

### 6.5.1. Transfer As\_RESULT:

- When a solid's geometry is transferred then the specification of the solid is not editable. It will however be possible to perform operations such as creating fillets, Booleans and Generative Drafting.
- Faces/Surfaces are always transferred as RESULT and converted into a V5 Skin in an Open-Body.
- With V424R1 a V4 IUA macro /chktop allows a topology check to visualize potential problems, such as non-joined faces, thin faces or too sharp edges.

### 6.5.2. Transfer As\_SPEC:

For the transfer of a solid's specifications, the restrictions in V5R11 are as follows:

- Certain types of primitives will be converted as datums:
  - Pyramids
  - Sweeps with a non-close profile, until a sweep
  - Sweeps with reference plane or reference surface and sweep with non planar center curve (transfer As\_SPEC of these primitives in the previous versions was an important cause of error)
  - Projects
- Following operation requires a complete transfer as geometry without history:
  - Keep Edge Draft
    - **Remark:** Being a smart solid is not a prerequisite to migrate a SolidE to V5 but fillet, draft or thickness specifications can be migrated more precisely if the solid is smart.
- In the case of “smart solids”, if the solid contains one of the above types of primitives, the following happens:
  - All supported elements are transferred, including CSG information.
  - Unsupported primitives are transferred as “Brep” only.
  - The V5 body is then based on both specifications and datum features.
- In the case of “non smart solids”, if the solid contains one of the above types of primitives, the following happens:
  - If the solid contains unsupported operations or primitives, nothing will be created. A warning message will be issued.
  - If all the primitives and operations are supported, they will all be transferred with the CSG information. The resulting V5 body is based exclusively on specification features.
  - Since the local results are not available, if the update meets some problems with contextual features, reroute help cannot be provided.

- 
- Sewing migration supports the “Intersection” and “No Intersection“ modes. By default, intersection mode is active. “No Intersection” mode can be activated exporting the variable: SewingNoIntersect=1.
  - Sweep migration takes into account some options for V5 Rib (or Slot) creation:
    - Keep angle
    - Reference element (plane or surface).
  - “Integrated Paste and Update in one step” option performs the entire migration operation in one interaction.
    - This mode includes a step-by-step check of the local results and allows an early detection of conversion problems. Additionally an automatic reroute of broken features is done.
    - Step-by-Step option works only on V4 Smart Solids and stops after the first V5 update error.
    - This mode can be enabled through a setting in:  
Tools -> Options -> General -> Compatibility -> V4V5 Specs
  - In case of problems with solid transfer, please refer to the “V4 Integration Users Guide”, where some workarounds and methodology have been described
    - Updating the pasted specifications may lead to some problems, particularly concerning contextual features (fillets, chamfers, drafts ... ) that cannot recognize their geometrical support. When this happens, the V4 face, edge or vertex is displayed to help the manual reroute of the specification.
    - If several solids are migrated, main Part Body is not used. Every solid is converted into a single part body connected to the main branch of the specification tree.
    - Draft Both Sides and Draft Multi-neutral are transferred in one primitive.
    - Inactive primitives or unresolved features are not migrated.

## **6.6. Transfer of Kinematics Elements to V5**

- V4 Kinematics Sets can be transferred into V5 (V4 2D Drafting Kinematics Sets are not migrated).
- Only V4 Multi-Set kinematics are supported; Mono-Set Kinematics have to be converted in V4 into multi-sets by using CATKI utility.
- All 3D joints are supported.

## **6.7. Transfer of V4 graphic properties to V5**

- V4 colors are transferred on V5 copied elements.

- 
- By default, elements in no show are not migrated due to performance reasons (except SolidE or SolidM in As\_SPEC migration). By exporting the KEEP\_HIDDEN\_ELEMENT=1 variable, a migration can be forced.
  - V4 layer numbers are propagated through data migration. They can be modified in V5 and used in V5 filter functionality.
  - Migration of V4 Filters to V5 can be done via batch migration job.

## **6.8. Transfer of V4 2D Textures to V5**

V4 2D textures can be copied and applied onto a V5 Material.

## **6.9. Transfer of User-Defined Elements to V5**

Most user-defined elements (UGEs and UAEs) cannot be transferred. These capabilities will become available when the dedicated application will be available.

## **6.10. Migration of V4 Standards (PRJ information i.e. patterns, text & annotations, etc.) to V5**

No plan. Has to be defined in Tools -> Standards.

## **6.11. Other Information**

- When an error occurs during migration, a diagnosis feature is attached to the V5 result. It gives access to an error message explaining the problem and localizes the problem in the 3D viewer.
- During the V4 to V5 migration a topology cleaner is applied automatically:
  - Some validity checks and cleaning actions are done (e.g. eliminates small edges, collapse confused edges, solves some configurations of too large gaps between adjacent faces, corrects inconsistencies of face or edge orientation).
  - It is possible to set a geometric tolerance to control the maximum gap allowed by these cleaning actions. The default value is “V4 Identical Curve Tolerance” (set V5V4CleanTolerance=0.01).

---

## 7 Batch Conversion of V4 Documents

**CATIA V5 offers a batch utility that allows the migration of a set of V4 documents.**

Note that the batch utility does not inherit of the ‘V4 family’, and ‘no show’ optimization offered in the Copy Paste mechanism.

### 7.1. Infrastructure support

- Until R10, each geometrical set in V4 was considered to be a V4 Part. Therefore, each geometrical set resulted in a V5 CATPart document. With R11, that behavior can be changed; alternately each V4 Solid will result in a V5 CATPart document (definition in Tools -> Options).
- Workspaces containing several sets or dittos are converted into a CATProduct. The dittos are converted as Product instances and reference the conversion result of the detail. Nested detail structures are supported and will generate “deep” (more than one level) Product/Part structures.
- \*.asm (V4 assemblies) are transferred as V5 CATProduct referencing the models (geometry is not migrated). V4 assembly constraints are converted into V5 constraints.
- \*.session “as reference” documents are converted as V5 CATProducts; the referenced models are also converted into V5 data format.
- Therefore the conversion result of a model can be either a CATPart or a CATProduct/CATPart structure depending on the model’s content.
- Batch migration tool is able to convert DRAW data. For each model containing DRAW data, a CATDrawing document will be generated; the DRAFTs will be converted as sheets, and the 2D details as detail sheets. Optional a Startup Model can be used for migration (definition in Tools -> Options -> Compatibility -> Migration Batch).
- The infrastructure settings in Tools -> Options -> Compatibility -> Migration Batch allows you to specify if only SPACE transfer, DRAW transfer or both are executed.
- MML Solids are migrated in As\_SPEC mode. MML Solids can be migrated in one shot (all \*.models have to be selected) or by using several submissions of CATV4toV5. Doing the migration in several steps allows a V5 repair of an imported component before the import of the other components of the MML structure. The pointing model must be updated when the pointing model is migrated; if not, no link will be created and the migrated solid will be a datum in the pointing part.
- Filter/Layer structure of a V4 3D model is converted into V5. A limitation to migrate only geometry laying on a specific layer (like in interactive copy/paste) is not supported in batch.

- 
- V4 Harness application data is migrated into V5 (Geometrical Bundles, Bundle Segments, Connectors). Some types of connectors are/will not be supported (see documentation). Electrical links (temporary restriction) and mechanical links (permanent restriction) are not supported as well as the migration of Electrical Component Libraries. Migrating Electrical Libraries is not planned; components can be simply inserted in a \*.model and converted to V5 where it can be used for the creation of a Catalog.

## **7.2. Batch submission syntax:**

Start from CATUTIL (Batch Monitor)

- MigrateV4toV5 (batch with graphic input window)
  - UNIX: `catstart -run CATUTIL -name MigrateV4toV5`
  - Windows: `CATUTIL -NAME MigrateV4toV5`
- MigrateV4toV5 (batch without graphic input window)
  - UNIX: `catstart -run CATUTIL -name MigrateV4toV5 <input-file>.xml`
  - Windows: `CATUTIL -NAME MigrateV4toV5 <input-file>.xml`
- CATV4toV5Migration (batch with graphic input window):
  - UNIX: `catstart -run "CNEXT -batch -e CATV4ToV5Migration"`
  - Windows: `CNEXT -batch -e CATV4ToV5Migration`
- CATV4toV5 (pure batch job; supposed to be used as trigger in VPM/LCA)
  - UNIX: `catstart -run CATV4toV5 .....`
  - Windows: `CATV4toV5`

## **8 Using V4 Models directly in V5**

### **8.1. Generative Drafting on a V5 CATProduct made up of V4 Models and V5 Parts**

- Only exact solids or \*skins are taken into account in the V4 model.
- FD&T and PARAM3D data are ignored and cannot be extracted. There is no plan to change this.

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## 9 V5 to V4 Conversion

### 9.1. Save a V5 CATPart as a V4 model

**It is possible to save a V5 CATPart as a V4 model. This capability increases the users productivity when integrating V5 data into V4:**

- Only CATIA V5 is necessary to access this functionality (no V4 license is necessary).
- The transfer of V5 data to V4 is the same as when using a File Open and Model/Copy of a CATPart in V4.
- Model Standards can be specified in two ways:
  - V4/V5 Infrastructure options can be used to specify some standards that will be used for the generated V4 model:
    - The V4 code page (WRITING CODE PAGE)
    - The V4 Model Dimension: modifying this value affects the tolerances of the generated model. The tolerances will be computed from the specified model dimension according to V4 recommended values.
    - The V4 Model Unit.
  - All of the previous options can be replaced by a reference to a V4 Initial Model. The code page, model dimension and model unit will be read in this model, and the generated model will also keep all the standards specified in the initial model.  
NOTE: mainly because of the UNIX/Windows compatibility, CATIA V5 will not be able to handle initial models containing geometry, so the initial models used in V4 may not be usable in V5, and a specific one may have to be created. If the initial model contains data that cannot be managed by V5 Save As Model, an error message will be displayed.  
The initial model can also contain 3D standards (defined in V4 STANDARD -> SPAC ELT -> SPEC ELT function) and layer/filters.
- The colors and layers applied on the V5 geometry are transferred on the V4 resulting geometry.
- Since there is no natural color mapping between V4 and V5, the colors transferred in V5 can be slightly different from native V5 colors. Using similar color maps in V4 and V5 should minimize this restriction.
- V5 feature names are reported as V4 identifiers.
  - Non-C2 continuous sets of faces are simplified into a single face whenever possible. This functionality is particularly useful when the V5 geometry itself has been created using pasted V4 geometry.
  - Whenever possible, Save As Model will fill in gaps larger than V4 tolerance. Such cases are not related to problems in V5 or in the V4 translation, but simply to a different tolerance concept between the two versions. They used to produce an “impossible merge” warning.
- When saving a CATPart as a V4 model, a report file will be created. If a problem occurs, the features involved will be specified to make problem analysis easier.

- 
- Activating the variable “V5V4SaveAsVolume=1”, forces Save\_As\_Model to create \*vol instead of \*sol.
  - Save As Model can be performed through a batch command. The batch mode proposes the standard Save As Model behavior (non associative), but also an **Associative mode**. In associative mode, V5 Part Bodies and Wireframe elements are supported.. The resulting model can be synchronized if the original CATPart is modified. This new mode allows the design modifications in V5 and the propagation to V4 downstream applications.
    - After updating the V5 CATPart, the previous version of wireframe elements are migrated in a user defined layer, the new version is migrated in current layer. That allows a manual reroute of V4 data that could be linked to this data.
    - Batch submission syntax: CATV5ToV4 -arg
  - Product description Property of the CATPart is migrated as Model Comment.
  - Topology cleaning during V5 to V4 migration:
    - An automatic topology cleaner is executed. The maximum gap value for these cleaning actions can be defined through the variable “V5V4CleanTolerance=0.01”. It’s the same variable as used in V4 to V5 migration.
    - In complement of previous behavior, small surfaces can be concatenated with large adjacent surfaces (if possible). Activation through variable “V5V4MaxSimplif=1”.
  - In case of a migration error, an error feature is created which allows you to locate the error area easily and to find the root cause the problem. Activation through a setting in Tools -> Option -> Compatibility -> Saving As V4 Data.
  - When converting V5 data into V4 data, a permanent identification is written in the .model file which shows that the model is coming from V5. PTF APAR HD01504 has to be installed in CATIA V4.

## **9.2. Save V5 CATProduct as V4 \*.session document**

**It is possible to save a V5 CATProduct as a V4 .session document. This capability extends coverage of V4-V5 supplemental processes:**

- Only CATIA V5 is necessary (no V4 license is necessary).
- Save As Session generates a set of V4-only documents:
  - CATPart documents referenced by the product are saved as models in the same directory as the created Session
  - V4 .model documents referenced by the product are pointed directly by the Session. The instances and their positioning in the product are converted as instances of models (modelos).
  - The CATPart name should not be longer than 64 characters.

- 
- When using this functionality on Windows platform the use of DLNAMES is required.
    - DLNAME must be written in uppercase.
    - DLNAME may not be longer than 16 characters.

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## **10 Downstream Application Data**

### **10.1. Reading and Using Engineering Application Data**

Geometric data related to Engineering applications such as tubing, piping, schematics, structures, electricity 3D and AEC primitives can be visualized.

### **10.2. Reading and Using NC Part Programming Data Created by the PROGRAMMER Series**

Currently no plan

### **10.3. Reading and Using NC SET Data Created by NC3/NC5 or NCLathe**

Currently no plan.

### **10.4. Reading and Using CATIA CL or APT Data**

Currently no plan.

### **10.5. Reading and Using Kinematics Data**

Kinematics data can be read in the DMU Navigator in V5 by adding a representation to a DMU Navigator product structure and in this case you can use the command "test and try" to replay the kinematics.

### **10.6. Reading and Using Robotics Data**

As of today, there is no plan to migrate this data to CATIA V5.

### **10.7. Reading and Using Fitting & Space Analysis Data**

Support for this capability is not in the plan because the V5 Product provides a more user-friendly capability and hence improves productivity.

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## **10.8. Reading and Using GPS Data**

- Read and reuse GPS data is in the plan with a low priority.
- It is not possible to use V5 GPS function directly on V4 models. The V4 model must first be converted to V5 and the V5 GPS capabilities can then be used.

## **10.9. Reading and Using FEM Data**

Currently no plan.

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## **11 Archived Data / CDM-CDMA Data Access**

### **11.1. Reading V4 CATIA Models stored in map and fic Format**

This capability is not supported in V5. We expect CATIA files to be mapless.

### **11.2. Reading Data stored in CATEXP/CATDATA Format**

- A utility is available to extract models from sequential files generated through CATEXP or CATDATA. There are some restrictions to this utility:
  - Only models are extracted: libraries or project files are not extracted
  - External PRJ information is not managed.
- This utility provides a very easy way to read in V5 sequential files containing 3D data or models with internal Project file. The utility can be launched:
  - in a pure Batch mode  
    CATEXtractModelFromSequential *-arg*
  - through a dedicated interactive interface  
    CNEXT.exe *-batch -e ExtractModelFromSequential*
  - through CATUTIL batch

### **11.3. Reading V5 Documents in MVS/VM CATIA V417Rxx**

There is no plan to update CATIA V417 code in order to support V5 documents.

### **11.4. Reading V3 Data**

V3 data cannot be read directly into V5 because of a map and fic/don support.

### **11.5. Reading in the CDM Database**

- It is possible to select a V4 model from the CDM database in the context of Assembly when adding a representation to a component of the assembly.
- This capability is only available on UNIX and there is no plan to provide it on Windows.

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## **11.6. Reading CDM-A Assemblies**

- Interoperability is performed with the CDM-A PSN window, allowing to interoperate with both CDM-A and VPM1 products, and allowing to take advantage of all selection dialog of the PSN (such as filter by volume, attribute values, ...).
- There is no plan to provide this capability on Windows.

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## **12 Windows vs. UNIX Limitations**

### **12.1. Impact of Characters supported on UNIX but not supported on Windows**

- In V5, only the characters belonging to ISO646 are supported for names of V5 documents.
- There is still a set of nine characters not supported on Windows but supported on UNIX:
  - greater than sign >
  - less than sign <
  - asterisk \*
  - colon :
  - quotation mark "
  - question mark ?
  - backslash \
  - slash /
  - vertical bar |
- In V5, these characters are forbidden for file naming. The impact is that UNIX files created with V4 with file names that include any of the above characters cannot be read on V5 Windows without being renamed.

### **12.2. Specific Windows provided capabilities**

- OLE related capabilities are only available on Windows (Insert of an external document inside a Drawing, COPY/PASTE a V5 document to an others non CATIA application, ...).
- Visual Basic Macros editing and replaying capabilities are available on both Windows and UNIX.

### **12.3. Limitations on Windows compared to UNIX regarding accessing V4 3D Libraries**

- Access to V4 3D libraries files is possible on Windows through the usage of an http server. In the option panel you must specify the location of the library directory, and this information is used when an access to a library file is necessary. DS will not provide the http server (e.g. http://<library-dir>/\*MASTER.library).

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## 12.4. Using V4 models under Windows referencing an external PROJECT file without UNIX

- Project File directory cannot be copied natively on Windows: the table file names generally contain Windows forbidden characters. CATIA V5 now uses a character map to retrieve the tables under a different name:
  - “ character is replaced with string ‘\_Inch’
  - \* character is replaced with character ‘x’
  - Other forbidden NT ( / \ < > : ? | ) characters are replaced with character ‘\_’
  - Non standard ISO characters are replaced with character ‘\_’

Renaming the Project File tables according to these specifications will allow the referencing of a Project File stored on Windows.

You can use the following methodology in a shell command to copy your Project File on Windows:

```
# Duplicate your Project File directory on UNIX:
cp -r <native_V4_PRJ> <NT_compliant_PRJ>

# Go to the new directory
cd <NT_compliant_PRJ>

# Rename the table files adequately:
mv .TEXTCOT±:±DESCRIPTIONS .TEXTCOT__DESCRIPTIONS
mv .TEXTCOT±:±STANDARDS .TEXTCOT__STANDARDS
mv PATTERN±:±DESCRIPTIONS PATTERN__DESCRIPTIONS
mv PATTERN±:±IDENTIFICATEURS PATTERN__IDENTIFICATEURS
mv PATTERN±:±MOTIFS PATTERN__MOTIFS
# Perform the same conversion for all table names when needed

# Once all the tables have been renamed, you can transfer this
# directory on a NT
#
# workstation and reference it in your compatibility setting.
```

**Note:** DO NOT modify directly your V4 Project File tables. CATIA V4 would no longer recognize it!  
On UNIX, CATIA V5 is still able to retrieve Project File tables under their standard name.