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Five Recommendations for Managing Legacy CATIA Data

"CATIA V4 obsolescence still poses a major problem for many product data managers. This technical brief identifies five recommendations for reducing cost and quality issues associated with CATIA V4 to CATIA V5 conversion, validation, and reuse."

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Global Business Development
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Introduction

The retirement of CATIA V4 on December 31, 2015, marked the end of life of a technology that has revolutionized the product development processes of global manufacturers and ignited a paradigm shift in digital manufacturing. Even though the developer of CATIA V4 (Dassault Systemes) began hinting at CATIA V4's retirement as early as 2009, many organizations still face the challenge of preserving decades of intellectual property created in the CATIA V4 CAD system.

Now that the vendor has officially ended its support for CATIA V4, business issues such as regulatory compliance, program restarts, failed system hardware, program safety or warranty issues, long-term archival, and/or changes to supply chain strategy, continue to spark a sense of urgency for those companies that are still in need of a CATIA V4-to-V5 transition strategy. In most cases, the hardware and software used to support CATIA V4 is no longer available and many workers who may have been knowledgeable of, and familiar with, the system have either retired or moved on to other projects.

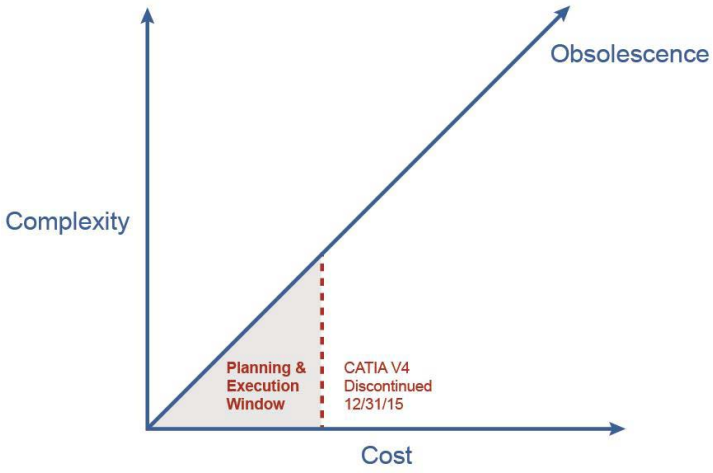


Figure 0.1: As the CATIA V4 obsolescence planning window narrows, cost and complexity increases.

Remastering existing CATIA V4 data so that it can be fully functional in the CATIA V5 environment is nothing short of complex, error-prone and expensive. Simply opening files can be challenging, and CATIA V4 possesses unique geometry entities (such as mock-up solids, DITTOs and piping entities) that pose a number of problems to the conversion process. Moving and maintaining history, features and drawings creates another challenge, while closing the quality gap through verification and validation is essential to building internal credibility, meeting regulatory requirements, and ensuring downstream reuse of legacy files and data.

Since 2008, consultants from International TechneGroup Incorporated (ITI) have successfully helped dozens of global organizations meet the technical and business challenges of moving CATIA V4 product data to CATIA V5. By combining flexible technology solutions, backed by a methodical, consultative approach, our team has been able to help our clients to deploy cost effective solutions while meeting 3D data quality and return-on-investment targets.

This technical brief offers five recommendations for IT managers and engineering stakeholders who are faced with the task of converting legacy CATIA V4 product data to CATIA V5:

1. Maintain a near-term solution for opening and viewing CATIA V4 models
2. Adopt an automated, CATIA-centric approach to editing CATIA V4 models
3. Upgrade CATIA V4 mock-up solids to exact CATIA V5 solids
4. Automate feature exchange, drawing associativity, and 3D MBD creation
5. Mitigate CATIA model divergence through CAD validation

1. Maintain a near-term solution for opening and viewing CATIA V4 models

An attractive alternative to migrating CAD data from CATIA V4 to CATIA V5 is to leave the data in its CATIA V4 form and make use of CATIA V5's ability to open and display CATIA V4 models and reference them within CATIA V5 assemblies. Unfortunately, many of the hardware and operating system technologies used to support CATIA V4 are retired and unsupported.

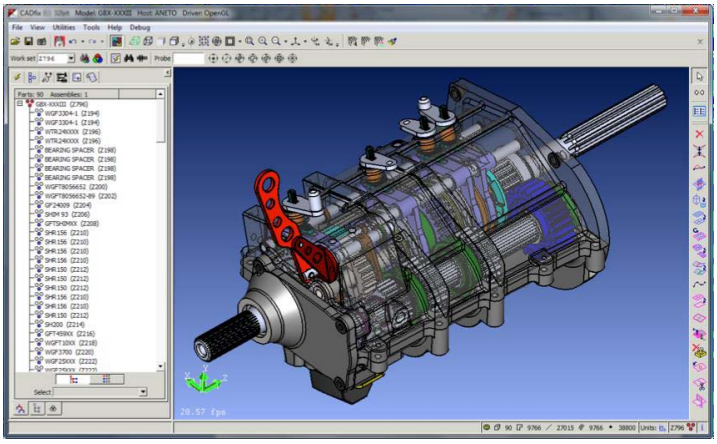


Figure 1.1 CATIA V4 model rendering within ITI's CADfix application

The first recommendation is to provide users with functionality that allows them to open and view existing CATIA V4 files without a CATIA V4 license. While users may lose feature and history information, the geometric entities will be preserved. ITI's CADfix solution supports a number of CATIA V4 element types and attributes, including SolidM, Pipe and DITTO assemblies.

Once the existing CATIA V4 files are opened in CADfix, users can identify and translate those entities that are vital to downstream reuse and save out the file to CATIA V5 (or any other CAD format). By avoiding the translation of additional or unwanted data entities, organizations will increase conversion success rates and realize significant return on investment (ROI) through labor savings. The screenshot provided in Figure 1 illustrates a CATIA V4 model displayed within ITI's CADfix application.

2. Adopt an automated CATIA-centric approach to editing CATIA V4 models

For some organizations, a mixed CATIA V4/V5 model format strategy is a cost-effective way to retain access to valuable legacy CATIA V4 intellectual property without the cost of recreating all of the product data. In this scenario, the ability to select and delete certain parts for redesign, while retaining the remainder of the CATIA V4 model intact, is a key capability for enabling the newly designed parts and original CATIA V4 data to be re-assembled in a mixed V4/V5 assembly.

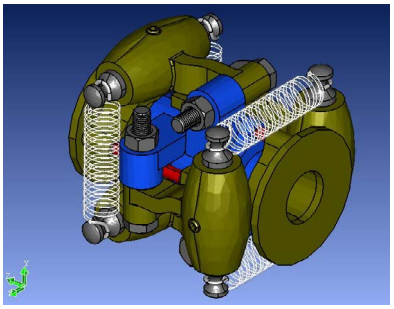


Figure 2.1: CATIA V4 model opened in CADfix, with selected parts deleted

The second strategy offers users the ability to open, edit and save CATIA V4 models containing a wide range of element types, including SolidM, Pipe and DITTO assemblies. SolidM support is offered in either converted BREP or original facetted forms; export of assemblies to CATIA V4 DITTOs is supported. An extended list of these capabilities is listed below:

- Import of SOLIDM entities in native CATIA V4 facetted form
- Display of SOLIDM entities within CADfix, so that they mirror the appearance in CATIA V4
- Export of SOLIDM entities to native CATIA V4 model file with full preservation of internal facet structure, appearance and compact storage
- Import of DITTO assemblies, including attributes (color, layer, label), on instances
- Export of an assembly to a CATIA V4 DITTO assembly, with instance attributes preserved
- Import of PIPE entities with conversion to exact BREP form (original pipe type held as attribute)
- Import of PIPE connect (CNP) entities as free points

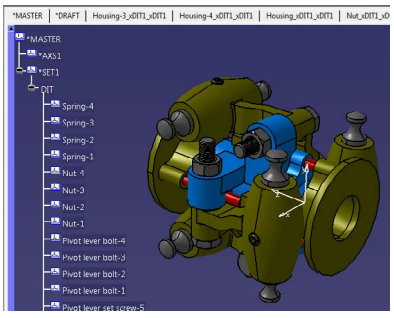


Figure 2.2: CATIA V4 model that was updated in CADfix now re-opened in CATIA V5

3. Upgrade CATIA V4 mock-up solids to exact CATIA V5 solids

CATIA V4 supports both an exact definition of a solid, and an approximate (or mock-up) definition of a solid. The mock-up form is a light-weight representation usually generated from an original exact solid, where both forms of the solid exist. Migration of the exact solid to CATIA V5 normally works well. However, mock-up solids can become isolated from their original exact solid, through certain operations within CATIA V4, and if the original exact solid is lost, the mock-up solid becomes the only existing definition of the part.

Migrating a mock-up solid to CATIA V5 does not result in an exact solid, but instead produces a facetted entity that performs very poorly in many common operations, such as visualization and drawing generation. To tackle this problem, a strategy is needed that can recover or upgrade a mock-up solid so that it relates back to its original exact solid form. The mock-up solid upgrade strategy requires a tool that is capable of analyzing the facetted nature of the mock-up solids, and re-generating the original smooth curve and surface geometry. Such an advanced capability is provided in ITI's CADfix solution.

The series of images below show the results of applying the CADfix mock-up solid upgrade tool to a typical CATIA V4 mock-up solid. Figures 3.1 and 3.2 show the facetted nature of the mock-up solids after import in to CADfix.

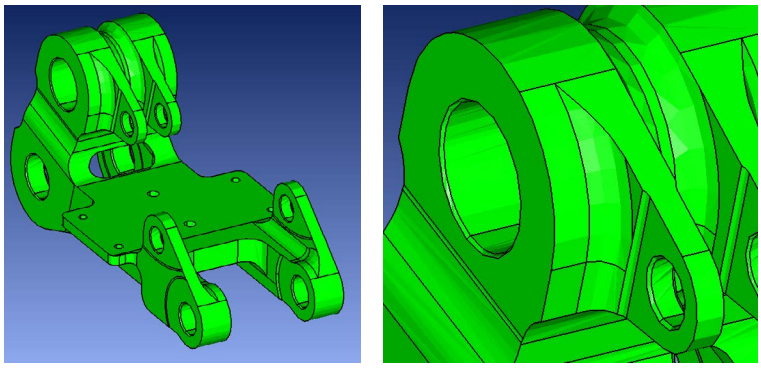


Figure 3.1 and 3.2: Facetted nature of mock-up solids after import into CADfix

The CADfix upgrade tool analyzes the facetted geometry and reconstructs the original smooth exact geometry. The "after" images below show the result of the upgrade, highlighting the regenerated curve and surface geometry. Figures 3.3 and 3.4 show some selected torus and cone surfaces recovered from the facetted mock-up.

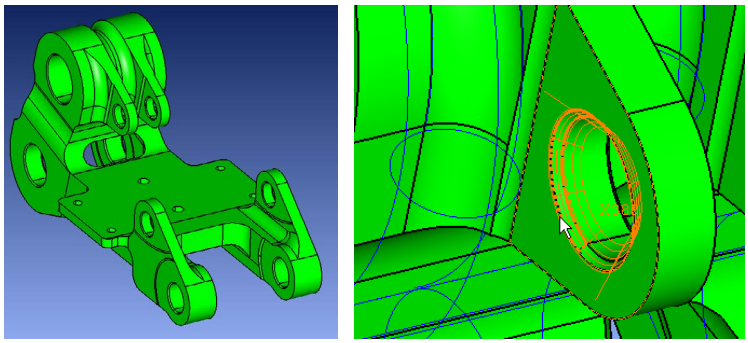


Figure 3.3 and 3.4: Right-hand image denotes selected torus and cone surfaces recovered from the facetted CATIA V4 mock-up using CADfix

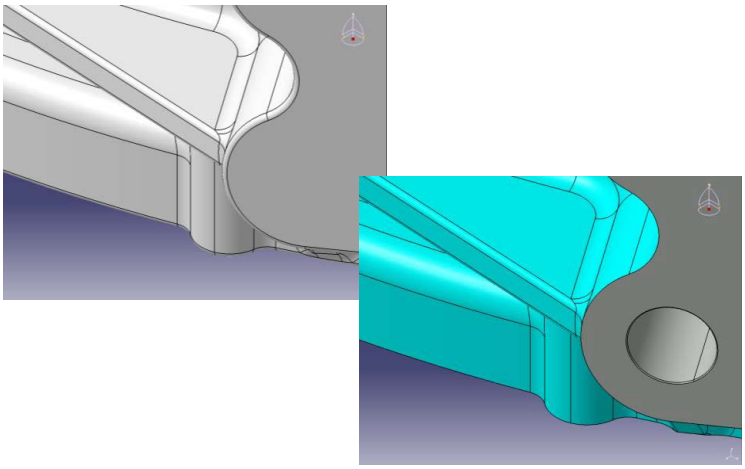
Once a mock-up solid has been upgraded to create an exact solid using CADfix, then it can be exported to CATIA V5 as a native CATPart model. The exact nature of the upgraded geometry enables the part to be treated as if it were an original CATIA V5 part— that is, the limitations seen with the faceted mock-up solid are completely removed. New modeling operations, such as cuts, blending, and more can then be applied. Figure 3.5 illustrates what the "upgraded" part looks like in CATIA V5.

4. *Automate feature exchange, drawing associativity, and 3D MBD creation*

Feature exchange between CATIA V4 and CATIA V5 is not a straightforward process. For instance, some features in CATIA V4 do not have feature equivalents in CATIA V5, some features are created differently, and some features fail to generate, due to geometry/topology requirements that are more restrictive in CATIA V5 than they were in CATIA V4.

ITI's Proficiency application uses a patented technology, called Feature Fallback, to identify incompatible features and replace them with accurate geometry. This technology's robust support for CATIA V4's multi-model-link (MML) structure enables the user to leverage a single automated process that includes structure and link mapping, the processing of non-synchronized and non-updated links, validation checks and reporting.

Figures 4.1 and 4.2 demonstrate how ITI's application produces conversions between CATIA V4 and CATIA V5 with MML support.



Figures 4.1 and 4.2: Missing geometry from an MML link (left) is successfully transferred using ITI's Proficiency solution (right)

In addition to filling these feature conversion gaps, Proficiency offers yet another patented technology for transforming 2D CATIA V4 drawing information in a 3D CATIA V5 model. This technology, trademarked as DrawtoPMI, converts CATIA V4 models with associative drawings and produces a CATIA V5 model with 3D PMI in CATIA V5, thereby automating a critical (and often manual) function, ensuring labor savings, functional data, and model accuracy. The illustration in Figure 4.3 shows an example of a legacy model and drawing produced in CATIA V4 that has been successfully converted to CATIA V5 using ITI's DrawtoPMI capability.

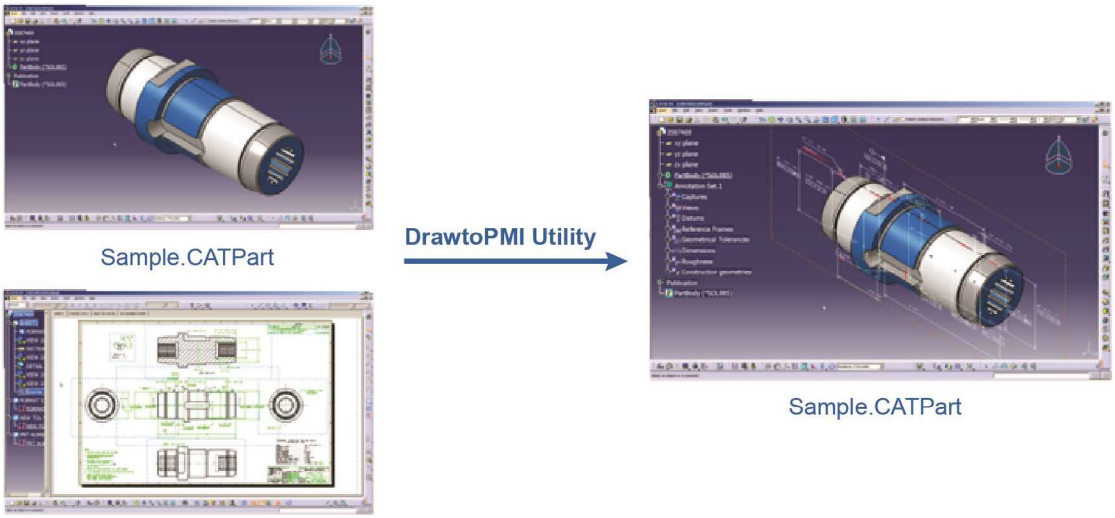


Figure 4.3: Using Proficiency to convert a CATIA V4 model with an associative drawing into a CATIA V5 model with embedded Product Manufacturing Information (PMI)

5. Mitigate CATIA model divergence through CAD validation

Each project has its own nuances, but there are essentially three factors for safeguarding the quality of your CATIA V4 data and then managing the risks of your CATIA V4 obsolescence initiative.

5.1 Assess the datasets prior to re-mastering or conversion

The functionality offered in ITI's technology, CADIQ, allows users to identify and segregate large groups of models that may not be candidates for automated conversion.

When these types of models or assemblies are identified, managers can use the Proficiency technology to automate a partial conversion, and then manually edit and repair models using the CATIA V5 Completion Wizard. Figure 5.1 below shows a legacy CATIA file with a geometry quality issue on the left. Proficiency, shown in the center, automatically completes the model and highlights where the errors were corrected. These hints allow the user to decide if they like Proficiency's automatic correction, of if they want to make further manual changes. The final model is shown on the right.

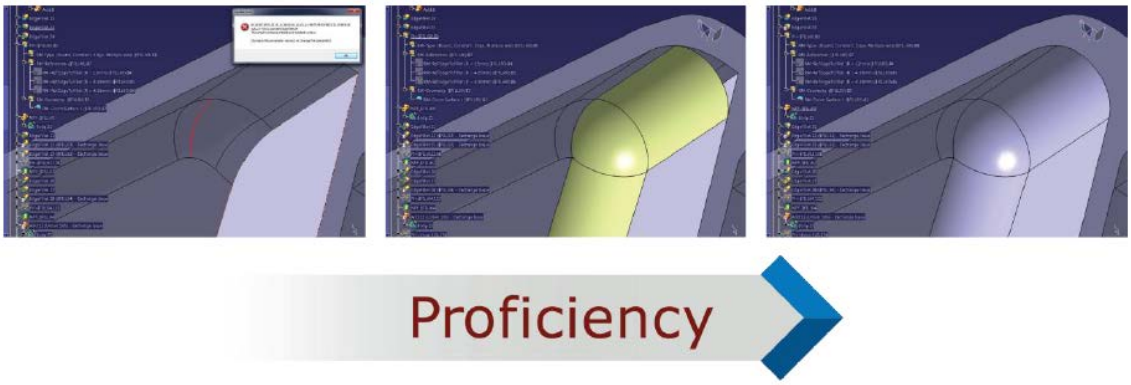


Figure 5.1: CATIA V4 model with missing geometry when imported directly into V5 (left) is repaired automatically using the Proficiency Completion Wizard (center). Proficiency automatically creates an accurate model that includes all legacy geometry (right).

5.2 Validate and certify the accuracy of the end result

Once the CATIA V4 data has been moved to CATIA V5, it is imperative that the end result in the target system be verified against the source. Through CADIQ, ITI supplies a vendor API-based solution for CAD validation. If there are discrepancies in the resulting output, the user can document those results by exporting a 3D PDF out of CADIQ to document the changes. Figure 5.2 shows a geometric/feature translation validation.

For semantic validation, ensure that the intent of the data has been properly translated, allowing for expected, insignificant variation. Figure 5.3 is an example of PMI translation validation.

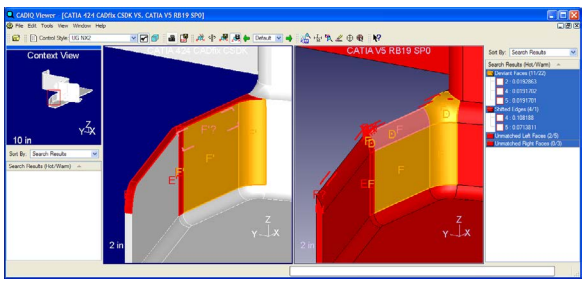


Figure 5.2: Example of geometric/features translation validation from CATIA V4 to V5

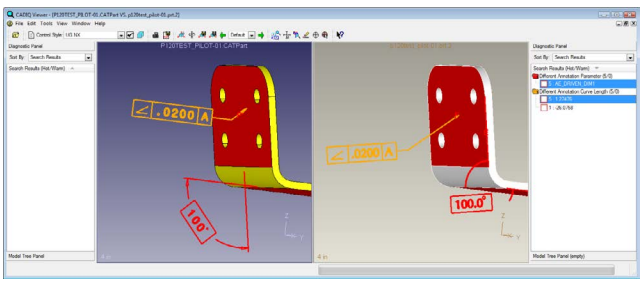


Figure 5.3: PMI translation validation in CATIA

5.3 Compare translation/re-mastering paths before selecting a provider

CADIQ also allows users to compare the output produced by any conversion tool or manual remastering provider. This functionality provides engineering managers with the power to evaluate vendor and third-party solutions side-by-side and determine the best tool or path for a given CATIA V4 dataset. Figure 5.4 is an example of a translator comparison used for a vendor translator trade study.

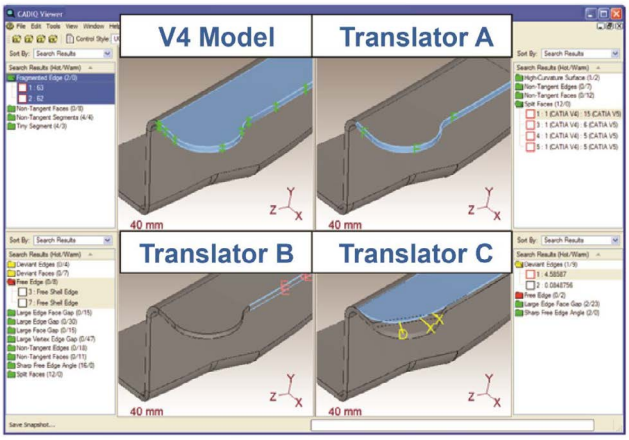


Figure 5.4: A translator comparison used for a vendor translator trade study

Conclusion

While updated versions of CATIA software have been introduced, legacy CATIA V4 product data still exists in many industries. To continue to use and maintain this data in CATIA V5 without access and support for CATIA V4 requires careful analysis of downstream user requirements and assurance that the CATIA V4 data is not only usable but accurate. When moving legacy CATIA V4 data to CATIA V5, solutions offered by International TechneGroup Incorporated (ITI) — CADfix, Proficiency, DrawtoPMI, and CADIQ — are ideally suited to: 1) reduce labor costs, 2) safeguard legacy intellectual property, and 3) ensure that the product data can be viewed, accessed, and reused in downstream systems and processes.

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About the Author

James Flerlage's publications have appeared in Desktop Engineering, Chemical Engineering, Windows IT Pro and syndicated business journals, such as the Boston Business Courier. His twenty years' experience spans both technical consulting and business roles. Flerlage currently serves International TechneGroup Incorporated (ITI) as Executive Vice President of Global Business Development. His credentials include a Master of Business Administration with Distinction from DeVry University and a Bachelor of Science in Telecommunications Management from DeVry University. To contact Flerlage, email james.flerlage@iti-global.com or connect via LinkedIn at https://www.linkedin.com/in/jamesflerlage.

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