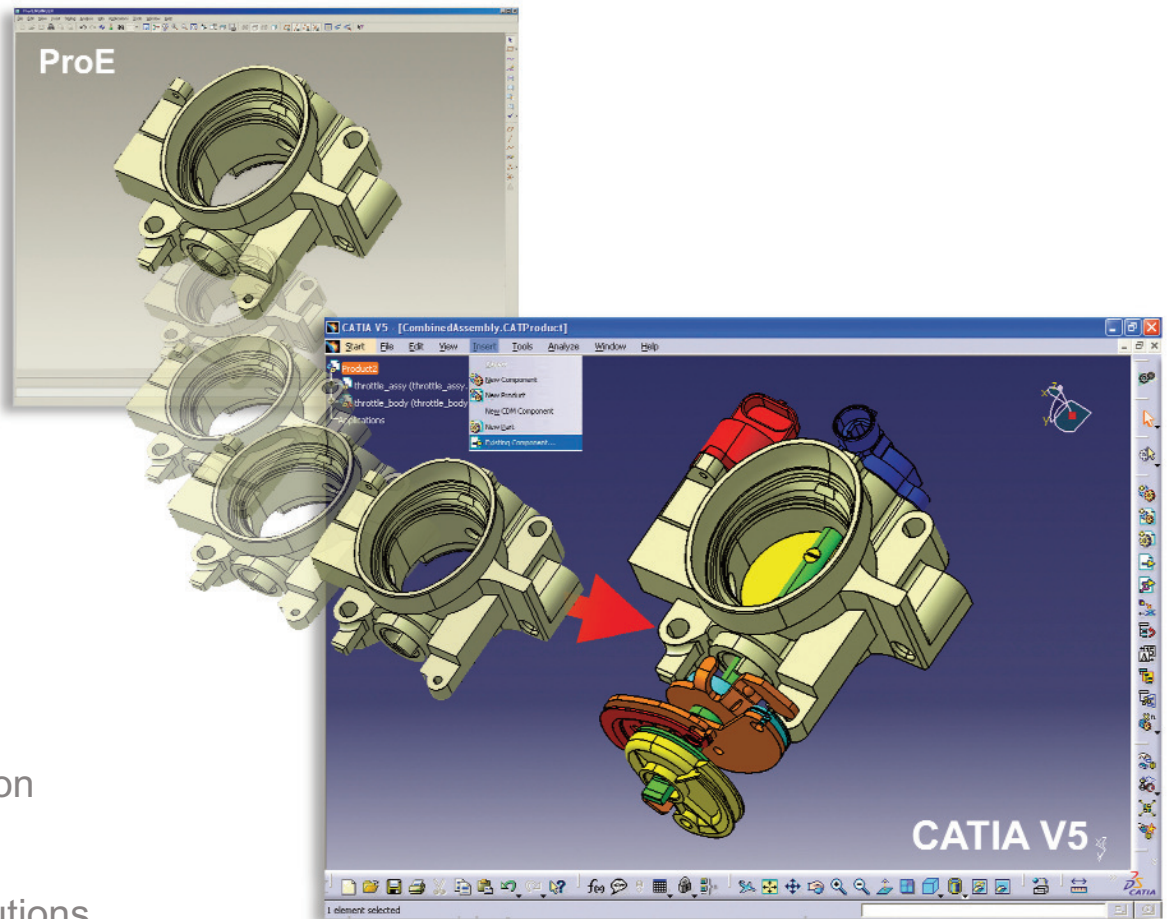


# CATIA V5 Multi-CAD Pro/ENGINEER Plugin



CAD

Visualization

STEP Solutions

Automation



**Theorem Solutions**

CAD Interoperability and  
Data Exchange

## The Challenge

Faced with the need to use both CATIA V5 and Pro/ENGINEER to complete a single product design and manufacturing task a designer has the choice of a number of different methods.

The traditional approach is to translate the Pro/ENGINEER data to CATIA V5, using either a standards based method such as STEP or a direct database translator. In doing so the original Pro/ENGINEER data is converted to a CATIA V5 representation. This task is either carried out by members of the design team or alternatively undertaken by dedicated translation specialists. There are several downsides to this approach. One is recognizing when a design change takes place in Pro/ENGINEER and managing the process of updating the CATIA V5 representation to take into account those changes once the initial translation process is completed. Another is having the knowledge to run a separate translation process and the time and costs associated with doing that.

## The Solution

Theorem's latest data translation products with advanced functionality are enabling the support of Dassault's interoperability strategy which places Multi-CAD in the hands of user, so that "design in context" can be undertaken without the need for external translation of non CATIA V5 parts and assemblies.

With Multi-CAD, CATIA V5 users are able using standard CATIA commands to select a part or assembly designed in Pro/ENGINEER, and insert it as a node on their CATIA V5 design tree; either as a single component or full assembly.

Selecting the required Pro/ENGINEER components to be worked on, automatically initiates the Theorem translation technology to bring the non CATIA data into the CATIA V5 session in the user's desired CGR or BREP format, depending on the CATIA session settings.

This highly interactive method of operation in V5 maintains and manages links with the imported files - enabling updated versions of them to be made

available to the user if the original Pro/ENGINEER part or assembly is modified. As a result, users benefit from the Theorem technology without needing to undertake a separate translation operation, and without having to worry if any changes have been made to the imported parts.

Once opened in CATIA, heterogeneous CAD data can be manipulated for Review and Design in context purposes whilst maintaining a link back to the original Pro/ENGINEER file.

## Additional features

Pro/ENGINEER Multi-CAD built upon the strategic Dassault Multi-CAD platform, enables the import of data to V5 with integrated control of data quality ensuring optimum CATIA data. Being XCAD based an option to output the data in 3DXML format is also available enabling its use in 3DVia and 3D Live.

Multi-CAD as well as enabling users to generate data that is linked back to the original Pro/ENGINEER representation, and therefore support the ability to track and update respective changes, will also support Design in Context DMU and explicit design modification. Using Multi-CAD with explicit modification enables the data to be subsequently modified within CATIA but removes all links back to the originating Pro/ENGINEER data.

Multi-CAD provides an option to cut the link to the referenced native Pro/ENGINEER data; Check and update is then not possible, but the Pro/ENGINEER data can then be modified within V5 using standard V5 explicit design modifications.

The application is also able to be run in batch mode outside of CATIA from an external command line interface. Running in batch mode it is possible to convert the CATIA V5 parts and assemblies back in to Pro/ENGINEER.

As standard Multi-CAD provides data in 3DXML format enabling the user to create a 3DXML light-weight representation of the NX data which can then be used within other Dassault applications, e.g. 3DLive, 3DVia.

## Highlights

- ▶ Interactive Design-In-Context solution for CATIA V5 users who need to access Pro/ENGINEER Parts or Assemblies.
- ▶ Maintains associativity between the native Pro/ENGINEER files and CATIA V5
- ▶ Enables automatic update of CATIA V5 representation when Pro/ENGINEER changes are undertaken.
- ▶ Uses standard CATIA V5 “Insert Existing Component” commands to introduce Pro/ENGINEER data into active CATIA V5 session.
- ▶ Built upon the strategic Dassault XCAD Multi-CAD platform, enables the import of data to V5 with integrated control of quality
- ▶ Using XCAD an option to output the data in 3DXML format is also available enabling its use in 3DVia and 3DLive
- ▶ Digital Mock-Up review (data imported with Faceted representation)
- ▶ Pro/ENGINEER import part is synchronized with CAD part or product structure modification.
- ▶ Update and synchronize V5 according to CAD data modification.
- ▶ Check consistency between V5 imported data and CAD data.

## Configuration options

The solution is packaged as:

Product Code	Description
PROMC5U	Pro/E to CATIA V5 uni-directional Multi-CAD
PROMC5B	Pro/E to CATIA V5 bi-directional Multi-CAD

A future optional capability in this translator to support 3D dimensions and annotation (FTA) import from Pro/ENGINEER into V5, any package not including FTA capability will be able to be upgraded to add this module.

## Prerequisites

- ▶ CATIA - MD1 (minimal pre-requisite), or any configuration including the Products of MD1, enables access to all product features except FTA. If a CATIA DN1, License is installed, the user will also benefit of the Standard DS CAT DMU Utility Batch Command. Access to FTA is enabled with a complementary license FR1 or FTA.
- ▶ DELMIA - any DELMIA configuration including the Products MDU or MNV + DOM, enables access to all product features except FTA. Access to FTA is enabled with a complementary license MFT.
- ▶ ENOVIA - ENOVIA DM1 (minimal pre-requisite), or any configuration including the Products of DM1 enables access to all product features except FTA. Access to FTA is enabled with a complementary license DT1.
- ▶ A license of Pro/ENGINEER must be installed on the seat of the customer

## Combinations of CATIA V5 & Pro/E Wildfire supported

Pro/ENGINEER	WF2	WF 3	WF 4		
CATIA V5 R18 SP5	✓	✓	✓		
CATIA V5 R19	X	✓	✓		

## Platforms

- ▶ Windows 32bit and 64bit

## Software License

Uses standard FLEXlm licensing technology

All product and company names and registered trademarks are the property of their respective owners and are duly acknowledged.



## Theorem Solutions

For USA and the Americas please use the following contact details:

**Theorem Solutions Inc**

6297 Tri-Ridge Boulevard, Suite 240

Loveland, Ohio, 45140-8396, USA

**Tel:** (513) 576 1100 **Fax:** (513) 576 1110

**Email:** [sales@theorem-usa.com](mailto:sales@theorem-usa.com)

For UK, Europe & Asia Pacific Region please use the following contact details:

**Theorem Solutions Ltd**

Tame House, Wellington Crescent, Fradley Park,

Lichfield, Staffordshire, WS15 4UP, UK

**Tel:** +44 (0) 1543 444455 **Fax:** +44 (0) 1543 444454

**Email:** [sales@theorem.co.uk](mailto:sales@theorem.co.uk)