# COLLABORATIVE DESIGN PROCEDURE USING CATIA V5 AND ENOVIA VPM

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*CATIA este o multi-platformă CAD / CAM / CAE - software comercial complet dezvoltat de compania franceză Dassault Systemes şi comercializat în întreaga lume de IBM. CATIA este utilizată pe scară largă în domeniul ingineriei industriale □i în special în aplica□iile auto şi aeronautică. În această lucrare vor fi prezentate unele proceduri, realizate cu ajutorul programelor "Macros" din Catia V5, care ajută la identificarea corectă a pieselor din interiorul unui ansamblu migrat de la Catia V4 la Catia V5 şi modificarea arborelui de comenzi pentru a putea fi vizualizate şi importate în ENOVIA VPM. Aceste programe macro-uri au fost dezvoltate (fiind necesare în cazurile practice) atunci când sunt necesare aplicații CAD dezvoltate într-un mod colaborativ între software-ul CATIA V5 şi ENOVIA VPM.*

*CATIA is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systems and marketed world-wide by IBM. CATIA is widely used throughout the engineering industry, especially in the automotive and aerospace domains. In this paper will be presented some programs, realized with the help of "Macros" of Catia V5, which helps "cleaning" the pieces inside a migrate ensemble from Catia V4 to Catia V5 and arranging the tree for visualizing and importing in Enovia VPM. These macros programs (necessary in practical cases) were developed when CAD applications use the software Catia V5 and Enovia VPM in a collaborative way.*

**Keywords:** Catia V4, Catia V5, Enovia VPM, ensemble, macro

## 1. Introduction

Computer Aided Three dimensional Interactive Application (CATIA) is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault Systemes and marketed worldwide by IBM.

CATIA is widely used throughout the engineering industry, especially in the automotive and aerospace domains.

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ENOVIA is the solution for informational management about design and fabrication of parts (ex. PDM II), included the industrial space modeling, configuration of digital models, caption and disseminated information and process knowledge (to the beginning of conception to sale and exploitation).

In VPM (Virtual Product Management), the VPM objects and their allocated product structure are graphically displayed via the Product Structure Navigator (PSN). The PSN is a graphical editor, in which the product structure tree is created and managed. In addition to this, the PSN offers further functions as for example the relative displacement of models/assemblies, query functions in connection with viewers, comparison of structures and relations of objects.

An assembly consists of parts and their models, which constructively form a group. An assembly represents a part of the product structure, in which it can be present multiple times.

In VPM, every part that itself contains parts with their models is an assembly. However, in general, there is no difference between parts with and without models. This means, that one can link a model to a part at any time (see Fig.1).

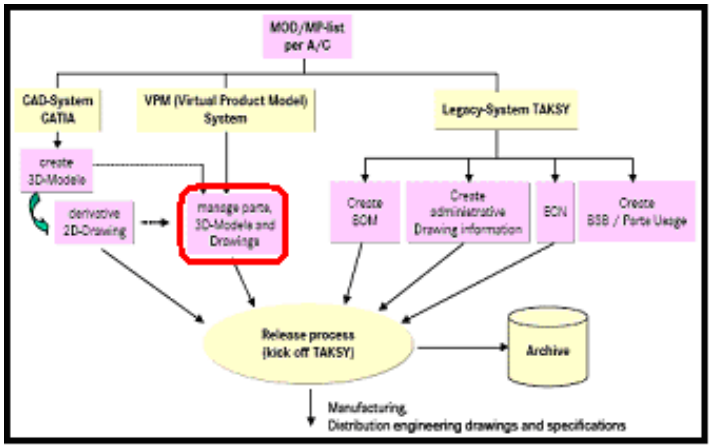


Fig. 1. Airbus Illustration of the Complete Process for Preparing Products

## 2. Macros

The objectives of this paper are to solve problems that occur from the transfer of parts and assemblies from CATIA V4 to CATIA V5 and the problem of import from CATIA V5 to ENOVIA VPM.

In this paper some examples of ensembles of pieces are presented, for which “Macro” instructions are used. These examples are typical for the transfer of Catia V4 products to Catia V5 products and the modification of the tree, necessary for it to be imported in Enovia VPM Software.

These macro programs were developed being necessary in the practical cases, when CAD applications use the software Catia V5 and Enovia VPM in a collaborative way (Fig.1).

The reason for which these programs were developed is the following: the Skylander projects usually use Enovia VPM and Catia V5 for CAD applications; the problem occurs in the moment when the transfer of the pieces and the ensembles is wanted from Catia V4 to Catia V5. The transfer is made as follows: Tools- Utility- MigrateV4ToV5 (Fig.2.). The result of migration is shown in Fig. 4.

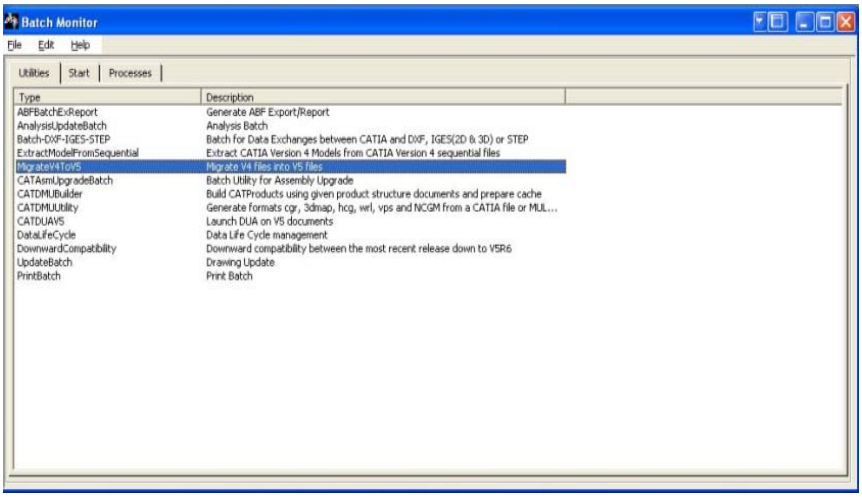


Fig. 2. Migration process in Catia V5

As a result of the transfer, there are pieces and ensembles, which have been wrongly named according to the initial naming of Catia V4, and this makes impossible the identification of the pieces (Fig. 4); thus they cannot be imported in Enovia VPM. The product representation in Enovia VPM is presented in Fig. 3.

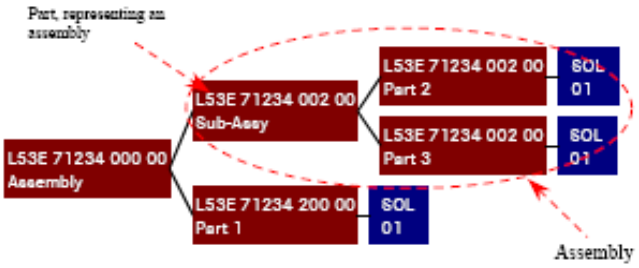


Fig. 3. Products representation in Enovia VPM

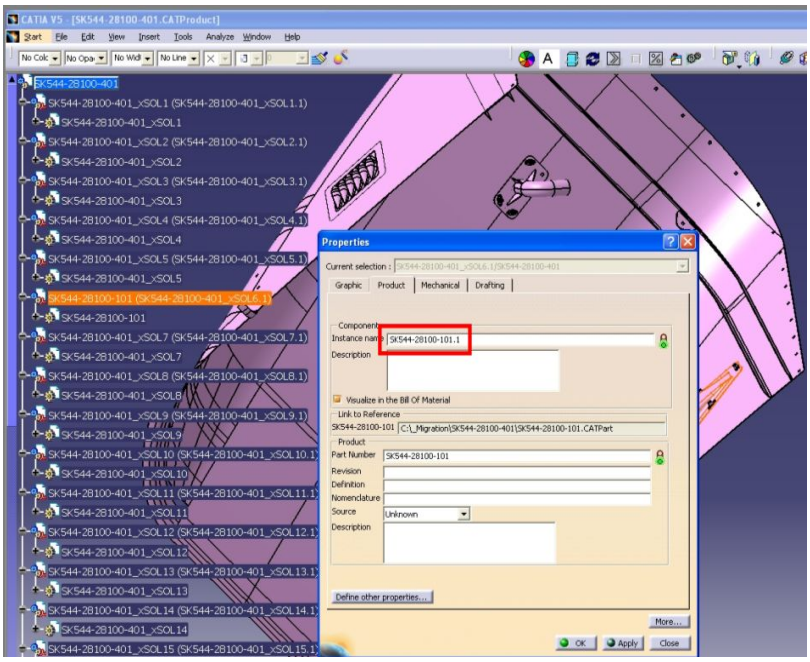


Fig. 4. Assembly after migration process.

The working procedure [4, 5, 6, 7] is the following:

- First, an appropriate name has to be given to the piece, and then the name must be stored;
- The Part Number must be like the file name;
- Useless elements must be erased so that in the end there should be only PartBody and GeometricalSet;
- The instance inside the ensemble should be changed too;
- The pieces inside the ensemble must be organized in an alphanumerical order;
- The final step is the creation of the STP file for import in Enovia VPM.

This procedure must be followed so the import in Enovia VPM can be done. This lasts very long time and there is a risk for making mistakes in naming of the files.

In order to reduce at its minimum the migration and the piece and ensembles correction time, the creation of macros that help at the correct renaming of the pieces and instances inside the ensemble was necessary [1, 2, 3, 8].

After the STP file is created, one can notice that the matrix for the piece positioning is not the same, and for this change the creation of some programs that will help the user not to reposition the pieces again in Enovia VPM was necessary..

To solve the problems that occur from the transfer of parts and assemblies from CATIA V4 to CATIA V5 and the problem of import from CATIA V5 to ENOVIA VPM, we created a series of macro programs that are listed below:

- **DeleteNonSolid.CATScript** - The macro will search the CATProduct and will select all the CATParts which have Non-Solid text string in the name of the CATPart file. To do the deletion one has to go in the menu Edit-Delete;
- **END_part.CATScript** - This macro will be used only for the CATParts coming from migration and will put the default name to PartBody, will write in the Properties Tab the name of the file, and will hide all non solids elements;
- **InstanceRenamer.CATScript** - This macro will rename the instances according with the CATPart or CATProduct name (see fig.5);

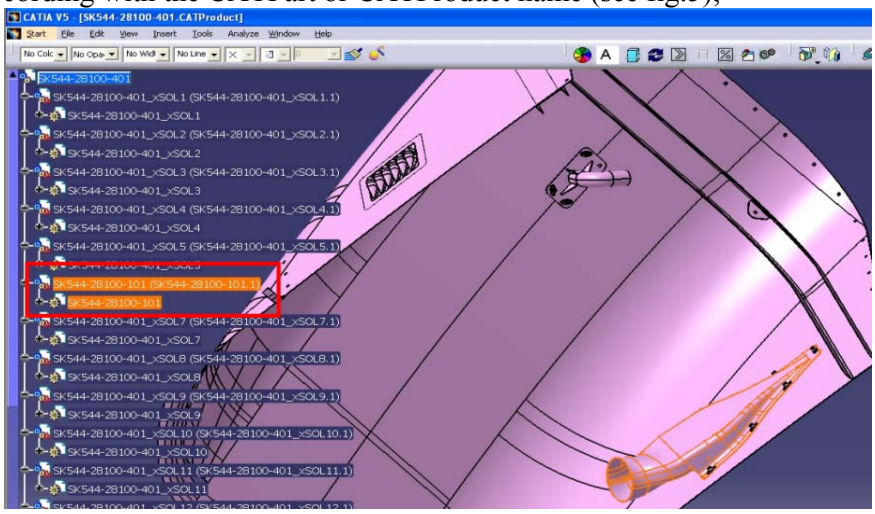


Fig. 5. The piece with the adjusted name and instance

- **PartBody_rename.CATScript** - The macro will rename the PartBody properties name to default (PartBody);
- **Reorder_Components.catvba** - This macro is working only under Windows OS, not in UNIX. The macro will reorder all the instances in the specification tree as you wish;
- **Rec_position.catvbs** - This macro will record the position of the selected items in the specification tree (same as in the Edit contextual menu of the compass). The positions will be recorded in an Excel file with the top product name but without any extension (see Fig. 7.-"Module 1");
- **Put_position.catvbs** - This macro will put the position of the selected items in the specification tree (same as in the Edit contextual menu of the compass). This is done using the recorded file by the Rec_position.CATScript. (see Fig. 7.-"Module 2").

The reason of the last two macros apparition is the following: the Airbus projects usually use Enovia VPM and Catia V5 for CAD applications; the problem occurs when large assemblies must be loaded from Enovia VPM into Catia V5. This problem appears during or after positioning of the assembly components when Enovia VPM is closed accidentally; in this case, the positions of the positioned parts in Catia V5 can't be saved in Enovia VPM.

If one intends to save this position only in Catia V5, this is possible, but this assembly cannot be opened with Enovia VPM in order to save this position after work. To this reason, it was necessary to conceive these two macros, one of them saves positions of components of the assembly in Excel files and the second macro makes the repositioning of the components without being necessary the positioning of them with specific instruments of the software Catia V5 using the module Assembly Design.

For this example, a small ensemble (11 components) was used, but this macros can be used for all assembles (independent of the number of components). Next, the position taken for "Part 2" element in Catia V5 with Parameter for Compass Manipulation help is presented.

In Figure 6, coordinates of the origin point (on top-left of the figure) and the angles (on top-right of the figure) for "Part 2" element are shown.

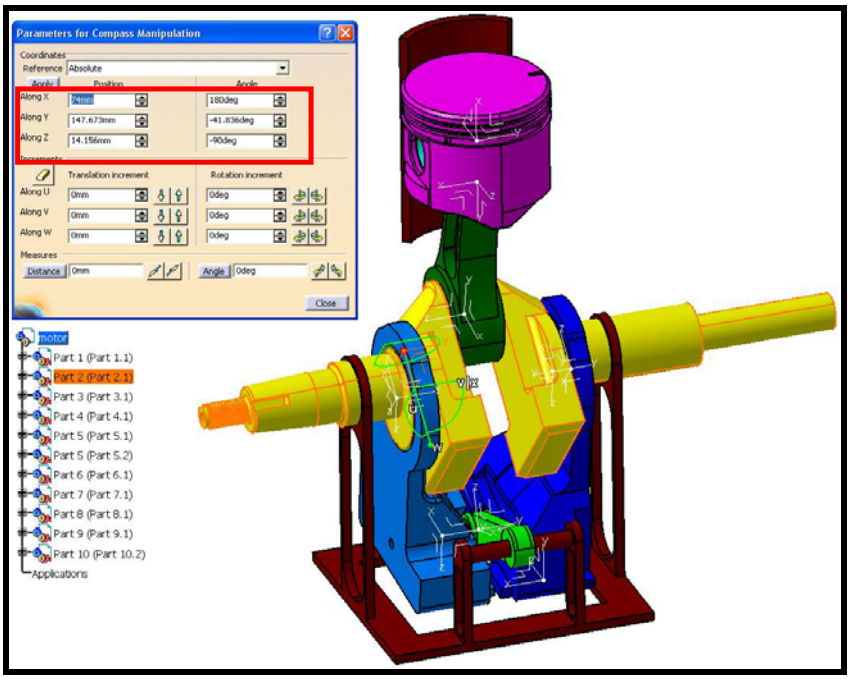


Fig. 6. Coordinates for "Part 2" element in CATIA V5

The use of macros to save the position of the ensemble components is made in Catia V5 with the following steps: Tools – Macro – Macros. After this command a new window appears, were one must select “Module 1” and then it gives the command “Run” (Fig.7).

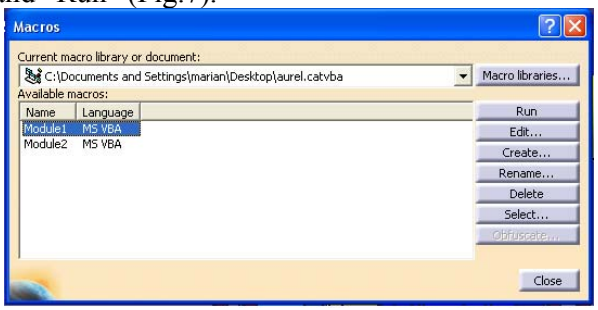


Fig. **7.** Macros from CATIA V5

These models read the products position and save in the Excel file. The file shall exist with the name: "C:\Temp\check_list.xls"; it can take a different name but the program line must be modified. After the module is executed, it must close the file for engaging the second module that read the save position and reload piece position in assembly. The files content is written in Microsoft Visual Basic.

In Fig. 8, some lines of macros ”Module 1” are presented; here one can see the position of the part as defined according to the absolute axis system of the assembly.

```
Set q = Excel.workbooks("check_list")
Set r = q.Sheets("sheet1")
Dim iAxisComponentsArray(11)
 ' x axis components
 iAxisComponentsArray(0) = 0
 iAxisComponentsArray(1) = 0
 iAxisComponentsArray(2) = 0
 ' y axis components
 iAxisComponentsArray(3) = 0
 iAxisComponentsArray(4) = 0
 iAxisComponentsArray(5) = 0
 ' z axis components
 iAxisComponentsArray(6) = 0
 iAxisComponentsArray(7) = 0
 iAxisComponentsArray(8) = 0
 ' origin point coordinates
 iAxisComponentsArray(9) = 0
 iAxisComponentsArray(10) = 0
 iAxisComponentsArray(11) = 0
```

Fig. 8. Module 1 representation in Microsoft Visual Basic

After the use of macros for the chosen example it was obtained the results seen in fig.8. In this figure we can see the representation angle by axes coordinate report on the reference assembly system.

In Fig. 9 the Excel file is presented. Here, the position of the part as defined according to the absolute system of axes of the assembly can be seen.

| Part number | Angle | | | | | | | | | Position | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | X | | | Y | | | Z | | | X | Y | Z |
| Part 1 | 1 | 0 | 0 | 0 | 1 | 0 | 0 | 0 | 1 | 0 | 0 | 0 |
| Part 2 | 1.46E-16 | -0.745063 | 0.666994 | -1 | -2.16E-18 | 2.16E-16 | -1.6E-16 | -0.666994 | -0.745063 | 74 | 147.6729 | 14.15619 |
| Part 3 | 2.79E-30 | 1 | 5.81E-17 | -1 | 2.79E-30 | 7.62E-17 | 7.62E-17 | 5.81E-17 | 1 | 35 | 279.9564 | -8.47E-14 |
| Part 4 | 1 | 5.81E-30 | 7.62E-17 | 5.78E-30 | -1 | 3.68E-16 | 7.62E-17 | -3.68E-16 | -1 | 35 | 250.9564 | -8.55E-14 |
| Part 5(instance 1) | 5.92E-17 | -0.745063 | 0.666994 | 1 | -2.2E-16 | -3.34E-16 | 3.96E-16 | 0.666994 | 0.745063 | -15.54 | 129.8378 | -5.766401 |
| Part 5(instance 2) | 3.96E-16 | 0.666994 | 0.745063 | -1 | 2.2E-16 | 3.34E-16 | 5.92E-17 | -0.745063 | 0.666994 | 85.54 | 129.8378 | -5.766401 |
| Part 6 | 2.03E-16 | 0.130298 | 0.991475 | -1 | 1.51E-16 | 1.85E-16 | -1.26E-16 | -0.991475 | 0.130298 | 85.54 | 129.8378 | -5.766401 |
| Part 7 | 6.93E-16 | 0.130298 | 0.991475 | -1 | 1.01E-16 | 6.86E-16 | -1.08E-17 | -0.991475 | 0.130298 | -5 | 129.8378 | -5.766401 |
| Part 8 | 9.81E-17 | 0.130298 | 0.991475 | -1 | 1.49E-16 | 7.94E-17 | -1.38E-16 | -0.991475 | 0.130298 | 56 | 34.11835 | 6.812848 |
| Part 9 | 8.48E-17 | 0.298309 | -0.954469 | -1 | 8.57E-17 | -6.21E-17 | 6.32E-17 | 0.954469 | 0.298309 | 35 | 34.11835 | 6.812848 |
| Part 10 | -1.08E-16 | -0.209356 | -0.977839 | -5.51E-17 | 0.977839 | -0.209356 | 1 | 3.13E-17 | -1.17E-16 | 35 | 157.6778 | 25.33213 |

Fig. 9. Excel file with position of the parts

## 3. Conclusion

The macros presented in this paper are very useful in practice because the designers do not waste time with renaming and cleaning parts throughout the assembly design. These macros could be a useful and very easy instrument for the new projects in the aerospace and automotive domain, if one inputs some modeling parts rules, similar with AIRBUS part experience. If these modeling rules are imposed and these macros are used, the necessary design time will be lower, and the cost will be about the same because only the text files are changing. In this type of procedure, the engagement of the environment for positioning these parts is necessary only for verification of possible interferences.

## R E F E R E N C E S

[1] *K. Amann*, Product lifecycle management: empowering the future of business: CIM Data, Inc.; 2002.

[2] *Mukundan Balaji*,- Design automation and KBE. Infosys; 2003.

[3] *J. Le Duigou, A. Bernard, N. Perry, J.C. Delplace*, A constraints driven Product Lifecycle Management framework. CIRP Design Conference 2009, 30-31 mars 2009, Cranfield, UK.

[4] *N. Predincea, C. Minciu, G., Constantin, M. Zapciu*, Precizia geometrică a maşinilor-unelte (Geometrical accuracy of machine tools) In: T.C.M.M. nr.11, Technical Edition House, Bucharest, 1995.

[5] *S.J. Schoonmaker,* The CAD guidebook: A basic manual for understanding and improving computer-aided design. Marcel Dekker: New York, 2003.

[6] *A. Szuder, , M. Zapciu, , C. Ispas, , T. Savu*, Bazele cercetării experimentale în ingineria mecanică (Fundamentals of experimental research in mechanical engineering), Technical Edition House, Bucharest, 1999.

[7] *M. Zapciu,* Cercetări teoretice şi experimentale privind deformaţiile elementelor portante ale maşinilor-unelte în vederea optimizării structurilor (Theoretical and experimental research on elements bearing deformations of machine tools to optimize structures). PhD Thesis, University POLITEHNICA of Bucharest. 1995

[8] *CATIA Version 5 Release 18 Documentation*.