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NEW MACROS TO ENABLE WORKING IN A COLLABORATIVE WAY WITH CATIA V5 AND ENOVIA

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Abstract: *To grow and stay competitive, companies in all industries must reduce time-to-market, improve product quality and lower the development costs. Traditionally, companies have managed product development in departmental silos, which has contributed to longer product lifecycles, delayed product launches and eventually loss of market share. In this work-paper a new feature based on two macros will be presented. The feature helps to saving the positioning of assembly components without saving in Catia V5 previously. These two macros programs were developed being necessary in the practical cases, when CAD applications use in a collaborative way the software Catia V5 and Enovia VPM.*

Key words: *Catia V5, Enovia VPM, ensemble, macro.*

1. INTRODUCTION

To grow and to stay competitive, companies in all industries must reduce time-to-market, improve product quality and lower development costs**.** Traditionally, companies have managed product development in departmental silos, which has contributed to longer product lifecycles, delayed product launches and eventually loss of market share [1].

To be competitive in the current world economy, companies need a PLM strategy centered on an integrated product model that incorporates manufacturing data. To enable such a system, the company needs a software infrastructure – a layer that interfaces with the operating system [6, 7].

An example of increasingly comprehensive PLM capabilities comes from Dassault Systemes (Suresnes, France), which has developed a PLM tool set consisting of three major components: CATIA for computer-aided design (CAD) and manufacturing (CAM); ENOVIA for PDM as well as communication and links for all the other tools; and DELMIA for manufacturing process management. Each product can work independently, but when they work together, they use relationships established in the design process to speed and ease the work of manufacturing and many other company departments [1, 3, 8].

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

CATIA V5 is a powerful tool to design and manipulate V5 objects and to work with V4 models.

ENOVIA is the solution for informational management of design and fabrication of the pieces (in the module PDM II), including the industrial space modeling, the configuration of digital models, the caption and disseminated information and the process knowledge (from the begin of conception up to the sell 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 that, the PSN offers further functions as e.g. the relative displacement of models/ assemblies, query functions in connection with viewers, comparison of the structures and relations of objects.

An assembly consists of parts and their models, which constructively form a group. An assembly represents 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. In general, there is however no difference between parts with and without models. This means, that you can link a model to a part at any time (see Fig.2).

2. MACROS

In this work-paper will be presented some examples of pieces assembly, for which "Macro" instructions were used. These examples are typical for the transfer of Catia V4 products to Catia V5 products and the modification of the tree, necessary to be imported in Enovia VPM Software [2, 4, 8].

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

The reason of the apparition of these programs is the following: the Skylander projects usually use for CAD applications Enovia VPM and Catia V5; the problem occurs in the moment in which 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.

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, thus they cannot be im-

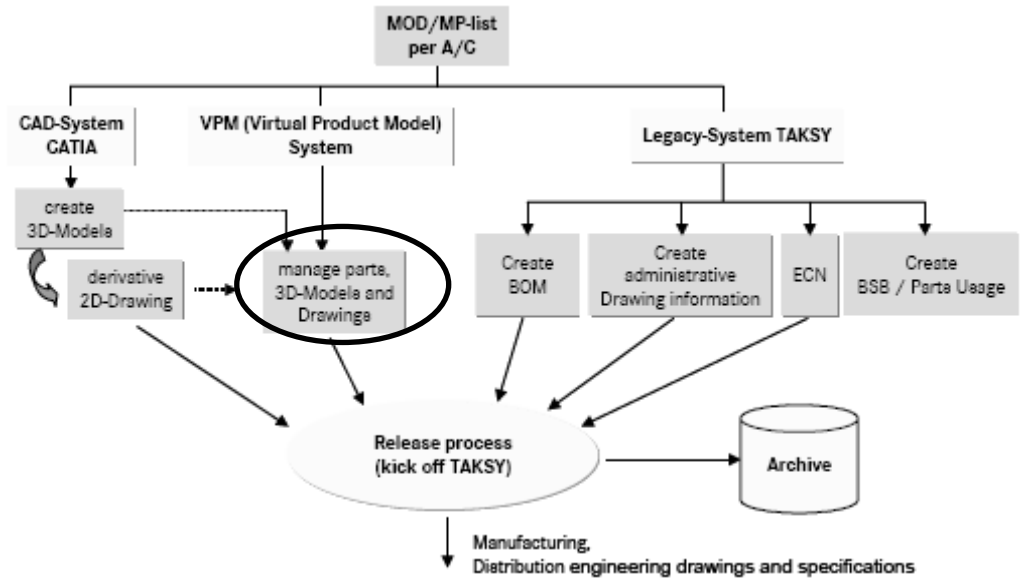

Fig. 1. Airbus illustration of the complete process for preparing products.

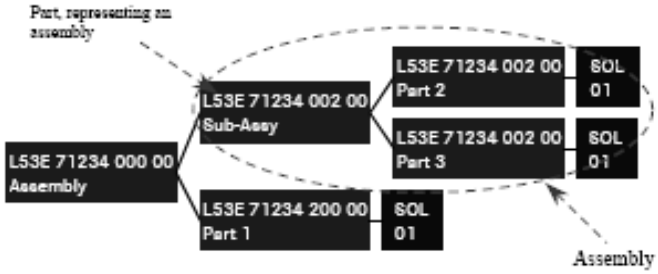

Fig. 2. Products representation in Enovia VPM.

ported in Enovia VPM. The product representation in Enovia VPM is presented in Fig. 2.

The working procedure is the following:

- firstly, 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 it can be imported in Enovia VPM.

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

In order to reduce at its minimum the migration and the piece and assembly correction time it has been necessary the creation of macros that help at the correct renaming of the pieces and instances inside the assembly.

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

These macros programs are presented down:

- **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 acording with the CATPart or CATProduct name.
- **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 items selected by you in the specification tree (same as in the Edit contextual menu of the compass). The positions will be recorded in a file with the top product name but without any extension (you can read it with a text editor).
- **Put_position.catvbs** – This macro will put the position of the items selected by you 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.

The last two macros programs will be presented below. The reason of the apparition of these programs is the following: a 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 cannot be saved in Enovia VPM.

If we intend to save this position only in Catia V5, this thing it possible, but this assembly cannot be open with Enovia VPM in order to save this position after work. For 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 other macro make the repositioning of the components without being necessary their positioning with specific instruments of the software Catia V5, by using the module Assembly Design.

For this example it was use a small assembly (11 components), but this macros can be used for all assembles (independent to the number of components).

Further on is presented the position take for the *vilebrequin* element in Catia V5, with the help of Parameter for Compass Manipulation**.**

The coordinates of the origin point (in the top-left of the figures) and the angles (in the top-right of the figures) for the *vilebrequin* element are presented in the Fig. 3.

To use macros that save the position of ensemble components it must be used this step in Catia V5: Tools – Macro – Macros. After this command it appears a new window where we must select *Module* 1 and then give the command *Run* (Fig. 4).

This models read the products position and save it in the Excel file. The file must exist and take the name: "C:\Temp\check_list.xls"; it can take a different name but the program line must be modified. After the execution, the module must close the file for engaging the second module who reads the saved position and reloads the piece position in the assembly. The file's content is written in Microsoft Visual Basic [2, 5, 8].

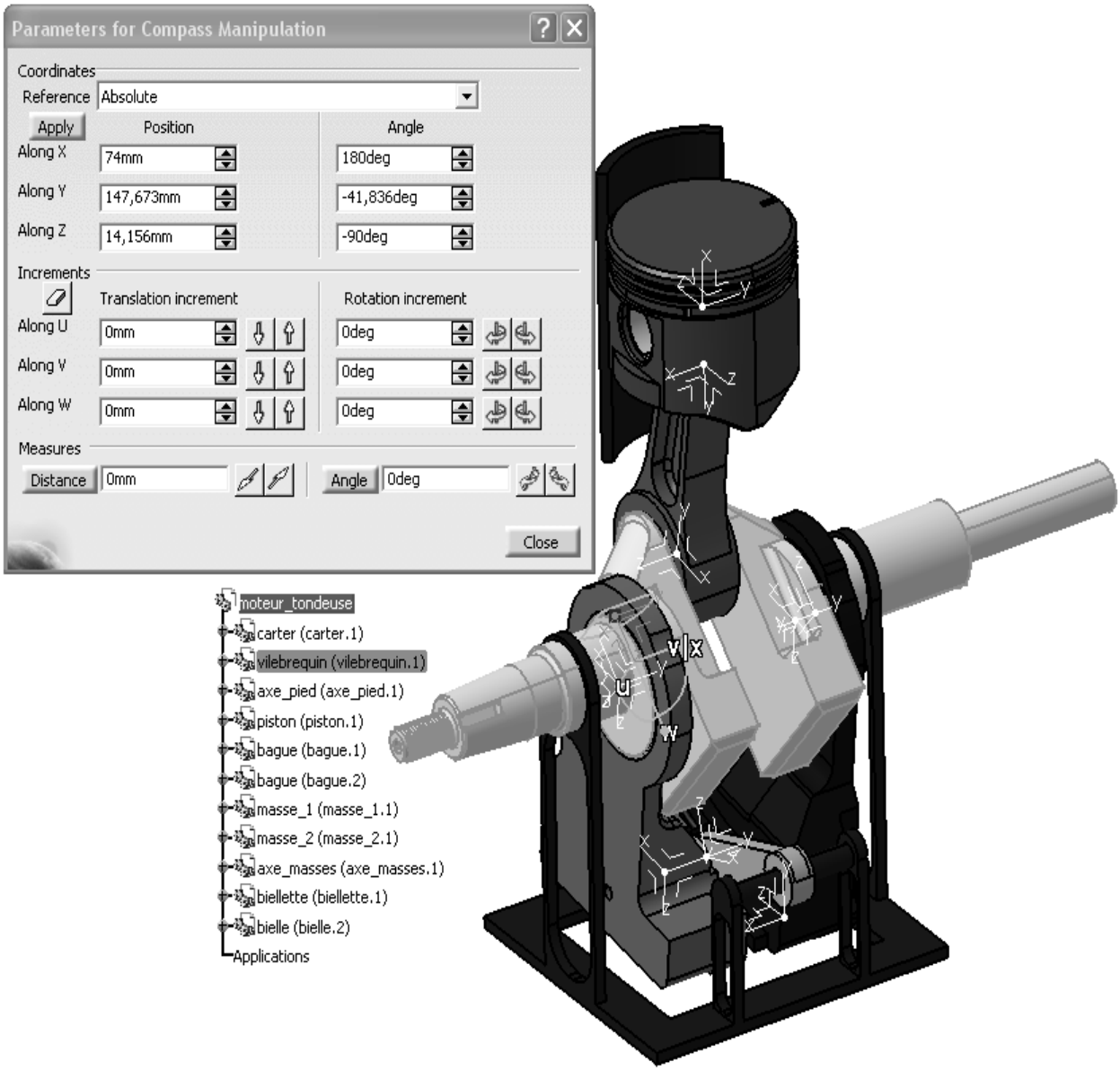

Fig. 3. Coordinates for *vilebrequin* element in CATIA V5.

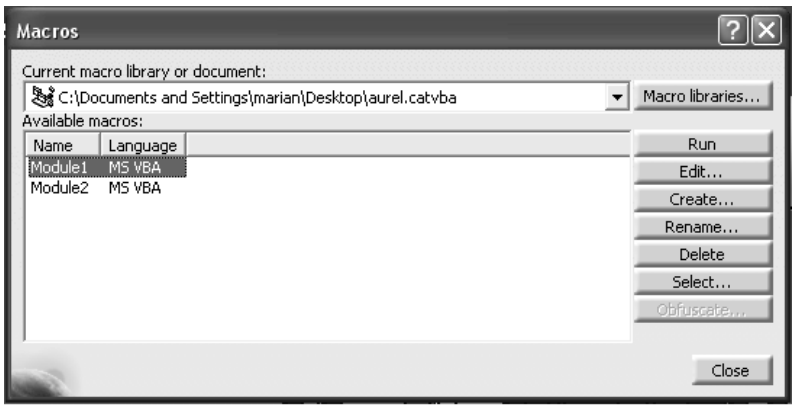

Fig. 4. Macros from CATIA V5.

Numele	Componentele axelor									Componentele originii punctului		
piesei	X			Y			Z			X	Y	Z
carter	1	0	0	0	1	0	0	0	1	0	0	0
vilebrequin	1,46E-16	-0,74506	0,666994	-1	-2,2E-18	2,16E-16	-1,6E-16	-0,66699	-0,74506	74	147,6729	14,15619
axe_pied	-2,8E-30	1	5,91E-17	-1	-2,8E-30	-7,6E-17	-7,6E-17	-5,9E-17	1	35	275,9964	-9,5E-14
piston	1	5,81E-30	7,62E-17	5,78E-30	-1	3,68E-16	7,62E-17	-3,7E-16	-1	35	250,9564	-8,6E-14
bague	5,92E-17	-0,74506	0,666994	1	-2,2E-16	-3,3E-16	3,96E-16	0,666994	0,745063	-15,54	129,8378	-5,7664
bague	3,96E-16	0,666994	0,745063	-1	2,2E-16	3,34E-16	5,92E-17	-0,74506	0,666994	85,54	129,8378	-5,7664
masse_1	2,03E-16	0,130298	0,991475	-1	1,51E-16	1,85E-16	-1,3E-16	-0,99147	0,130298	85,54	129,8378	-5,7664
masse_2	6,93E-16	0,130298	0,991475	-1	1,01E-16	6,86E-16	-1,1E-17	-0,99147	0,130298	-5	129,8378	-5,7664
axe_mass	9,81E-17	0,130298	0,991475	-1	1,49E-16	7,94E-17	-1,4E-16	-0,99147	0,130298	56	34,11835	6,812848
biellette	8,48E-17	0,298309	-0,95447	-1	8,57E-17	-6,2E-17	6,32E-17	0,954469	0,298309	35	34,11835	6,812848
bielle	-1,1E-16	-0,20936	-0,97784	-5,5E-17	0,977839	-0,20936	1	3,13E-17	-1,2E-16	35	157,6778	25,33213

Fig. 5. Excel file with origins of the pieces.

After the macros use, an example is chose in order to obtain the results in Fig. 5. In this figures we can observe the representation angle by axes coordination with regard to the assembly system reference.

3. CONCLUSIONS

In conclusion, these two macros are very useful in practice because no time is lost in with the repositioning of the pieces in the assembly's interior.

These two macros can be easily used in the new projects in the aero-spatial and the automotive domain; in the case these projects input some pieces-modeling rules, similar to the Airbus piece. If these modeling rules are implemented, the necessary work-force for this macros will be lower, and the costs will be the same because only old excel files must be changed, without being necessary the engage of the environment in this process. The environment's task will be to only verify the interferences.

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