Aspects on Transfer of Aided – Design Files

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Abstract. At this stage of development of hardware and software, each company that makes design software packages has a certain type of file created and customized in time to distinguish that company from its competitors. Thus today are widely known the DWG files belonging AutoCAD, IPT / IAM belonging to Inventor, PAR / ASM of Solid Edge's, PRT from the NX and so on. Behind every type of file there is a mathematical model which is common to more types of files. A specific aspect of the computer -aided design is that all softwares are working with both individual parts and assemblies , but their approach is different in that some use the same type of file both for each part and for the whole (PRT), while others use different types of files (IPT / IAM, PAR / ASM, etc.). Another aspect of the computer -aided design is to transfer files between different companies which use different software packages or even the same software package but in different versions. Each of these situations generates distinct issues. Thus, to solve the partial reading by a project different from the native one, transfer files of STEP and IGES type are used

1. Present day situation and terminology used

In order to clarify the present day situation, the terminology used in the field of information transfer associated to assisted design should be firstly clarified. Therefore, the term model represents the informatics reflection of the physical body (actual or imagined by the designer) and involves two components, one of the mathematical/geometrical type, and one of the alpha numerical type. The first component contains all lines, circles, ellipses and *Spline* curves that the programme creates connections amongst, according to the mathematical model which constitutes the basis of the software and which finally defines the three-dimensional geometrical form of the landmark. The physical model and the data file saved on the hard disk under different extensions should not be considered identical. The computer processor may recompose the physical model in the memory of the calculus system, by summing up the geometrical elements and the alpha numerical attributes only if the data file is read/opened exclusively by the software application that created it. In other situations, opening the files using other software packages is achieved partially or faultily. Regardless of the CAD type software package used in opening the files achieved natively in other design packages, it does not allow the restoration of parametrical drafts and of the 3D characteristics achieved based on them. In certain exceptional situations, as in the case of the Solid Edge software, there is the possibility of launching a manual reconstitution option for all modelled three-dimensional characteristics, but the procedure is laborious and takes a very long time. The ideal situation is based on the case when each one works individually on a project, in the same software, and the fabrication is achieved within the

same company. In reality, the situation is totally different from the ideal case because there is a large array of design software and an accentuated specialisation of companies in very narrow fields of study, which requires good collaboration between companies and a permanent information transfer by means of work files. There are numerous situations in which two or three design companies work together on the same project, but do not use the same design software package, or in which the designer works in one software, while the beneficiary uses another. Very often, there are situations in which the ensemble project transmitted to the beneficiary cannot be opened or totally read by the latter, even if the same design software package is used. In such situations, the essential problem is the loss of links between the ensemble-type file and the individual part-type files. In the case of multi-national companies capable of producing a integrated software for CAD, CAE and CAM, the transfer of data between the design modules is no longer necessary because integrated files are used, containing all the information about the 3D model (CAD), as well as the finite elements analysis (CAE), or the simulation of fabrication (CAM), including the administration of the product for the whole duration of its life span.

2. The transfer of files between various design platforms

With the development of technological necessities and of hardware support in the design sector, information transfer between companies that use various design packages became a must. The transfer was made on the one hand between various CAD applications for 3D design and, on the other hand, between the latter and CAE applications which analyse finite elements and simulate the resistance capacity of the products designed. In this sense, the interface of CAD products has historically evolved along two directions [1]:

- ✓ model information transfer through exchange files – called formats conversion;
- ✓ integration methods.

2.1. Conversion method

The first direction mentioned above is the most frequently used because especially designed exchange files have been generated, to be opened by the majority of design software packages, and it is defined by the existence of the following three formats:

- ✓ An original format, which is also called source format, of the application which created the model.
- ✓ An intermediate data CAD exchange format which, in the most common of situations, is named IGES, STEP or ACIS [2, 3, 4].
- ✓ A destination format or, more specifically, the format of the application in which the intermediary file, imported in the destination type application, is saved.
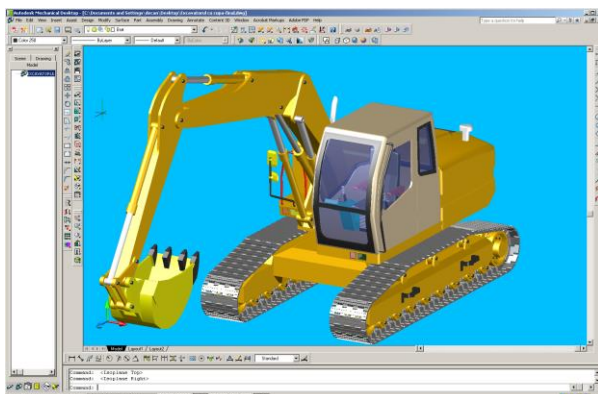
This method of transfer by means of intermediary files, called formats conversion, was a big step towards progress, in the sense that the import of transfer files was more practical than the remaking or regeneration of the model within the new application. However, it is characterized by the following inconveniencies:

- ✓ The accuracy of the model resulting from the conversion (compared to the original one) depends on how correct the conversion of the source data into the exchange format and of this data into the destination format is. Each of these two mandatory stages of the conversion is achieved through various programmes, written by various suppliers, programmes which do not always rally at the same time to the last specification/version of the exchange format.
- ✓ No conversions can be achieved with 100% efficiency (without any loss of information). If this loss means only that, for example, two lines congruent at the top inside the source application no longer part in the same spot in the destination application, it remains acceptable, but if the sense of a normal to surface has changed, then the respective face becomes invisible at a "lines hiding", or it is possible that the meshing/analysis with finite elements produces the behaviour of another body. Sometimes, the cleanup resulting from the conversions is as laborious as the
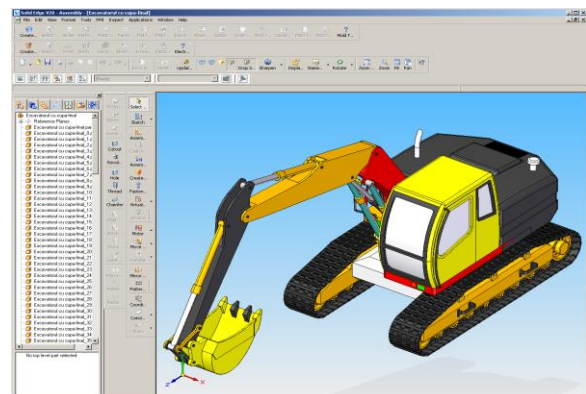
partial remaking of the model in the destination application and, certainly, takes a very long time.

✓ No exchange format has succeeded and will never succeed in keeping up with the growing diversity and complexity of information regarding the model to be transmitted between programmes.

✓ Until recently, the greatest inconvenient of the formats conversion method consisted in the fact that a dynamic connection, which can update the model automatically (or on request) in one/both application(s) depending on the modifications carried out in each, could be established between the source model and the destination model (or the respective files). Disconnecting the "source" files from the "destination" files because of the intermediate file presupposes the fulfilment of a system (manually in most cases) of administering the modifications and the file versions (versioning), to signal the models moments of synchronization.
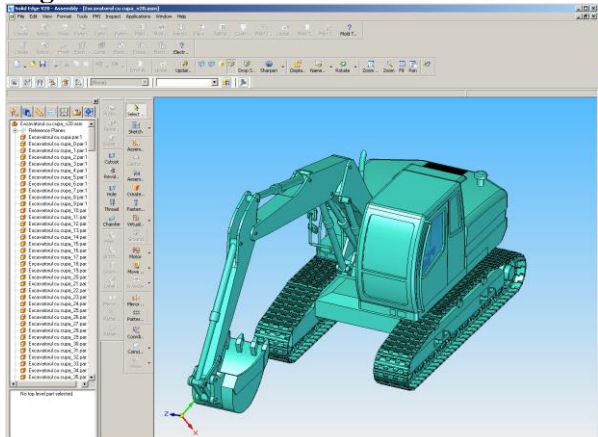
Therefore, Figure 1 represents an excavator achieved in AutoCAD 2008, opened with Mechanical Desktop 2008, which was exported in multiple exchange file versions, namely IGES, STEP or ACIS. The result of the import of these exchange files in Solid Edge v20 according to the documentation valid at that moment [5, 6], is presented in Figures 2, 3 and 4.
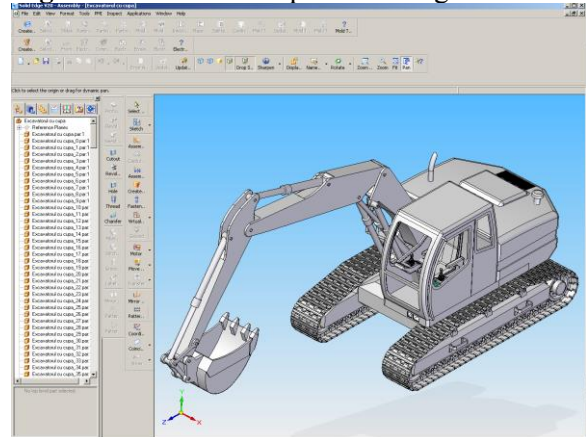

Figure 1. Excavator achieved in AutoCAD 2008.

Figure 2. Excavator imported through IGES files

Figure 3. Excavator imported through STEP files

Figure 4. Excavator imported through ACIS files

2.1. Integration method

This method was the best for an important period of time and was based on files integrated and achieved by the same production company. If the conversion method is a method of interfacing between programmes, the integration technologies represent, in fact, methods of interface between models.

The essential difference from the formats conversion method consists in the fact that the data does not alter its native format. The conversion method (especially the one based on objects) ensures the coexistence of methods arising from different source applications in the same destination application, without the import/export modifying the rapid format of the data. The best known case of the integration method is ACIS modelling which, in fact, represents a mathematic formalism for representing solid bodies and the relations between them, independently from any CAD programme. Having in view its performances, as well as the fact that it is developed by a neutral company (Spatial Technology Inc., who do not produce CAD software) ACID modelling has become a "de facto" standard for solids modelling and for the transfer of these models between applications. Although the use of ACIS exchange models practically means import/export of data by means of an intermediary *.sat file, this method differs in principle from the classical conversion method in that both the source application and the destination application must internally implement the same mathematical formalism (algorithm licensed by Spatial Technology) to save/restore the model in/out of the *.sat file. An application which implements ACIS does not only know how to write or read *.sat files by translating its own internal format, but encapsulates a strict mathematical scaffolding to define and manipulate solid bodies, based on the specifications emitted by Spatial Technology.

Market studies show that ACIS is and will remain the most widespread integration method among CAD programmes specialised in solids modelling; today, adopting a CAD programme of solid modelling not rallied to ACIS forces the user of the respective product to be "isolated" on an informatics island (even if it is very efficient, as, for example, Pro/ENGINEERING from Paramedic Technology or Euclid from Matra Datavision).
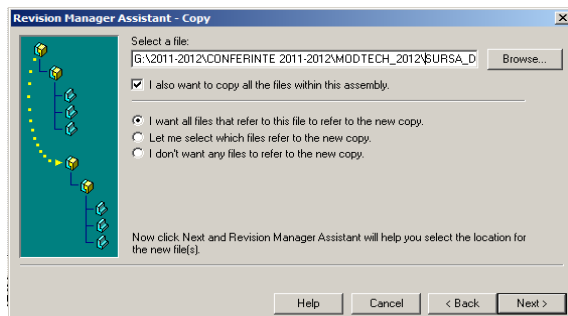
ACIS must be differentiated from the technology of objects, even if it is part of the same category of methods, because:
- ✓ Integration through ACIS models remains a method which is dependent on the product (specifications) and the destiny of a company.
- ✓ ACIS remains a method of integration/transfer of graphical objects, being restricted to the transfer between CAD applications, exclusively for geometrical information. The ACIS methods cannot be integrated, for example, into an application which calculates materials quotes or consumptions and which can "explore" the model through known interfaces in order to find the attributes necessary for calculus.
- ✓ ACIS is, firstly, an "engine" for modelling solid bodies, and is so widespread that it has became a data exchange "vector", without being initially intended for this purpose.
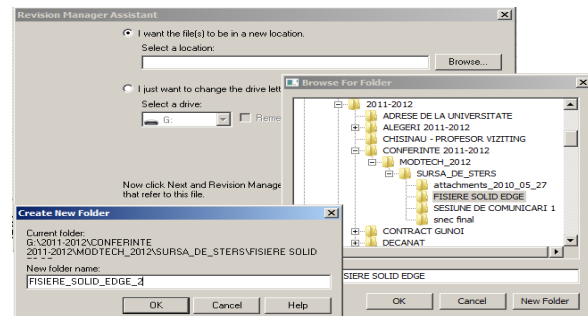
The objects technology represents a much more comprehensive way of integration between CAD applications on the one hand, and between these applications and the universe of non-CAD applications on the other hand.

3. Transfer of ensemble-type files between the same CAD type software packages

Reality demonstrates that, in certain situations, the ensemble-type files generated with design software in a certain specialised company cannot be opened successfully in beneficiary companies which use the same version of design software. This is due to the fact that in some situations the links to "part" type files are lost in the three-dimensional ensemble. As a result, each software producer has its own personal way to avoid these deficiencies. Therefore, in the case of the Inventor software created by Autodesk, the secret lies in launching the "Projects" command (which defines the place where all the designed components will be saved in view of archiving the whole project to the beneficiary) before the start of a work session. By contrast, Solid Edge has as alternative method of generating a clone of the project in view of its being transmitted to the beneficiary. This method is presented below, based on a project which had as purpose the achievement of a de-multiplying sequential mechanism. The stages covered were the following [7]: launching *View and Markup*, saving changes made in the current file, order launching *Revision Manager*, launching *Revision Manager Asistant*, launching *Perform Actions*, *Save Actions* and *Close View and Markup*.
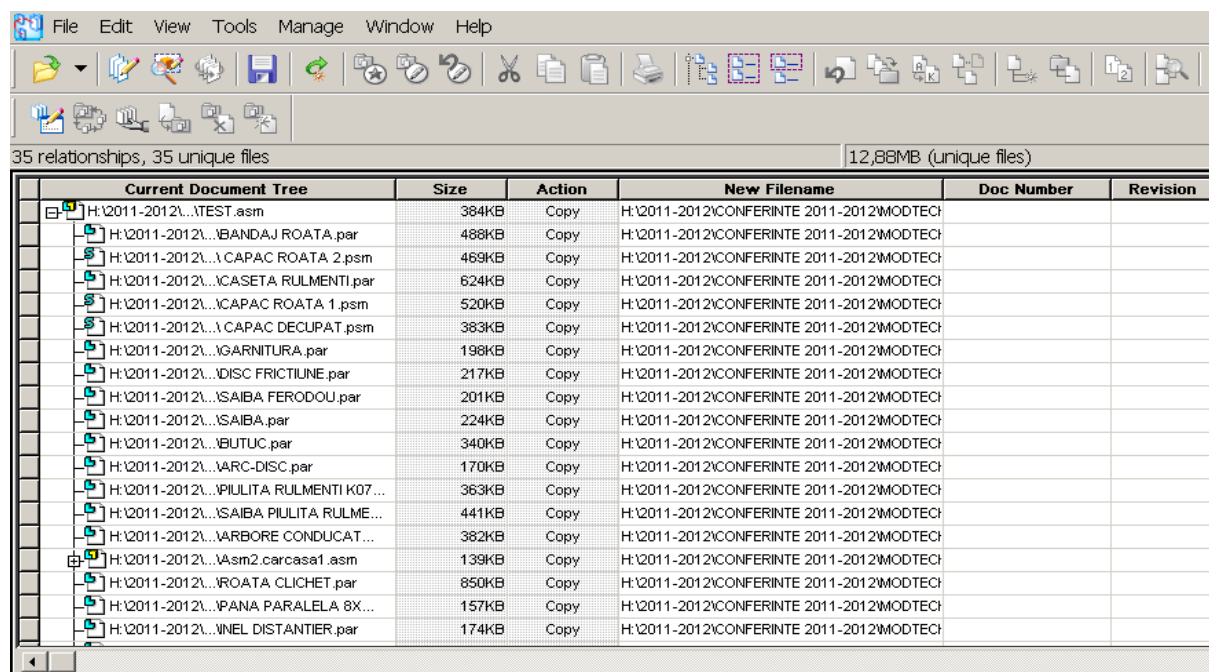
Figure 5. Choosing the ensemble-type file.

Figure 6. Defining the place for storing the clone.

From all the stages presented above, the most ample is the Revision Manager Assistant launching stage, within which the following sub-stages must be followed. The first sub-stage is choosing the type of action which refers to copying a file in view of cloning, revising the file or searching a certain file. For the present study, the variant of copying in view of cloning was chosen. A second sub-stage consists in choosing the ensemble-type file exactly as in Figure 5 and defining a new directory according to Figure 6, in which the obtained clone will be stored.

Figure 7. The result of cloning the files.

As may be observed from Figure 7, the clone obtained can be totally analysed and used on other computers also. In other words, moving the project in another PC, which determines another arborescent positioning of the project in the destination PC, does not affect the visibility of and the access to the project. In conclusion, this new method presented here eliminates the disadvantages of the classical move of a project from one directory to another by simply copying the files from the source directory into the destination directory. The classic method of copying determines the loss of connections between the standardised elements and the rest of the landmarks within the project, which implies the impossibility of visualising them in the 3D ensemble model. In contrast with the classic model, the method presented above is generally valid for all types of CAD projects developed with Solid Edge and allows the transfer of projects from one PC to another without losing the geometrical information associated to the standardised elements used in the project.

4. Files transfer through importing and preserving the link to the source file
This new transfer method belongs to the Autodesk Company and is specific to version 2016 of the design application called Inventor. The method mentioned above is called AnyCAD and solves a special problem which frequently occurs in achieving ensembles in Inventor by using various "part" type files achieved with other design software, like Solid Works, Pro Engineer, Catia, NX etc. Until now, Inventor could introduce in an ensemble-type file not only Inventor type parts, but also parts natively achieved in other design software applications, yet if one of them is modified in the native application, it must be reintroduced in the Inventor ensemble. If only one part was modified, there would be no problem, but in practical design situations the modifications are successive, for tens of parts out of the total of hundreds which constitute the respective ensemble. As a result, redefining the ensemble would last a very long time. The essence of the new AnyCAD concept consists in the fact that Inventor 2016 allows the insertion of natively achieved parts in other design software, and their direct modification in the native software automatically leads to immediate acquisition of the modifications at the level of the Inventor ensemble. In other words, AnyCAD is an integrated technology of interoperability which facilitates a direct and immediate association without the need for transfer files between the file natively generated in other software and the destination type project achieved with Inventor 2016. In order to exemplify, the landmark called *excitatory body* was taken into consideration and it was achieved by using Solid Works 2014 as integral part of a compacting board, which was imported in a project achieved with Inventor 2016 without using intermediate transfer files [8].
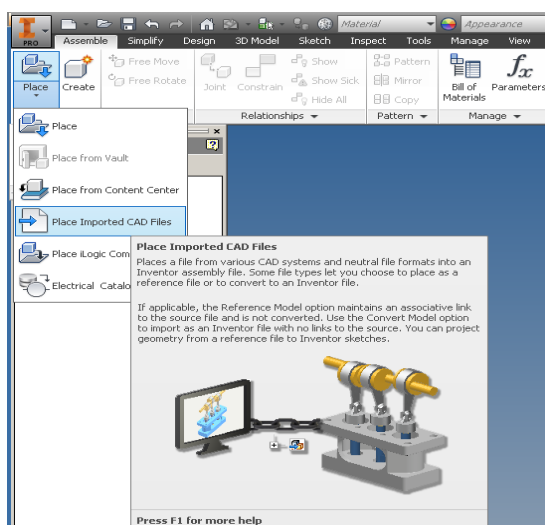

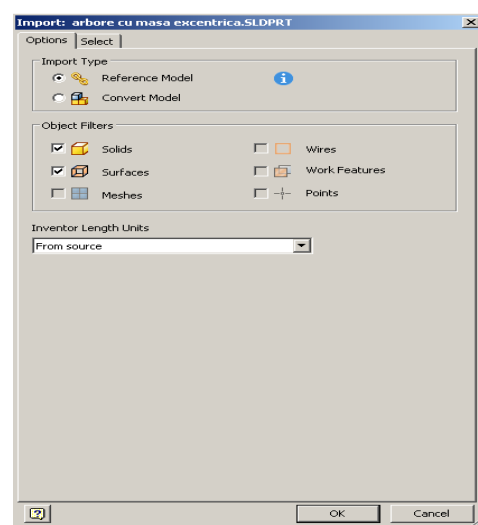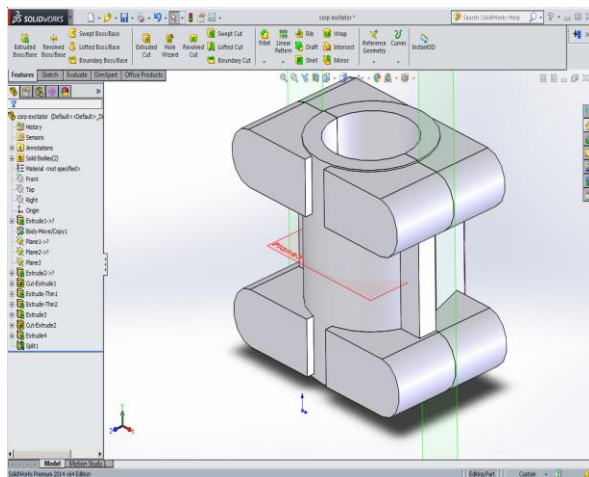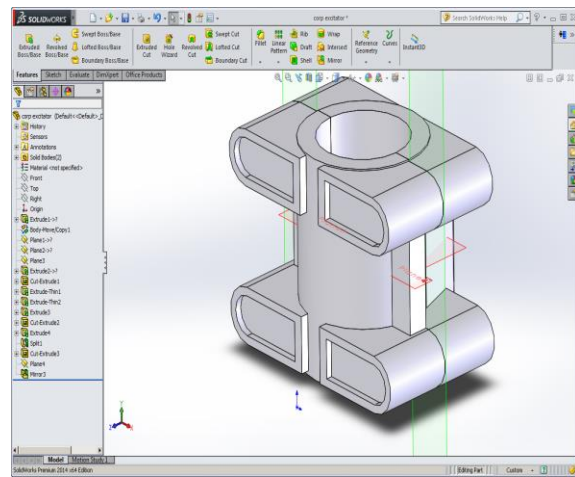**Figure 8.** Importing the Solid Works component.   **Figure 9.** Import settings.

Figures 8 and 9 present the dialogue windows used for importing in Inventor 2016 the file created in Solid Works 2014. The production company for this new technology of importing files in the Inventor environment did not communicate details about the importing mechanism, but what is clear to any user of Inventor 2016 is the fact that there is a FULL associativity between the model imported from Solid Works and the resulting model, with a *.iam extension, in Inventor 2016. This associativity is not restricted to the geometrical form, but also covers the alpha numerical information which defines the physical properties of the imported part. We can therefore speak about a transfer of intelligence between the native Solid Works type model and the Inventor 2016 type destination model. This even allows successfully performing a Nastran simulation in Inventor, to see if the significance of the whole project is maintained. This work technology with any CAD file suggests (starting with its AnyCAD denomination) the capacity to work with other very popular CAD applications without the need for remodelling or re-importing the files created by these applications. Nevertheless, there are also limitations, in the sense that this interoperability is limited to the following CAD applications:
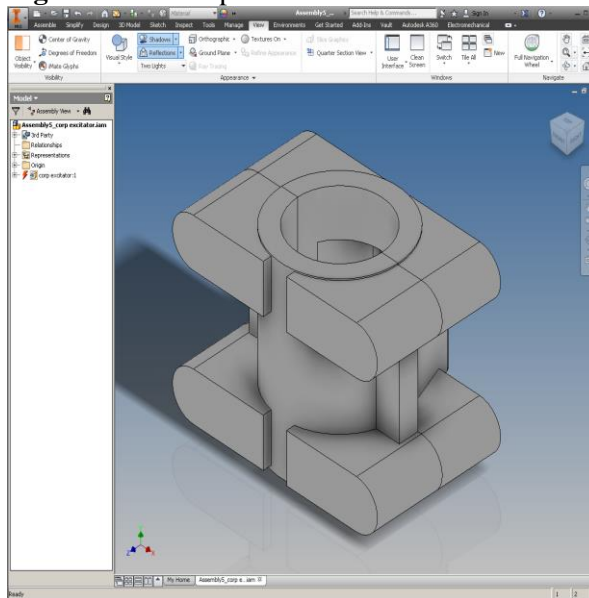
SOLIDWORKS (version 2011 Plus – 2015), PTC Creo (version 1.0 – 3.0), Autodesk Alias (version V10 and later), Catia (version VA and R6-V5 6R2014), NX (version UGV12-NX9), Pro/Engineer Wildfire (version up to Wildfire 5.0). The novelty of the AnyCAD application consists in the fact that it has two simultaneous characteristics: it creates a model converted in Inventor without the need to use intermediary transfer files, which in turn involves a total associativity with the reference model created by other design products.
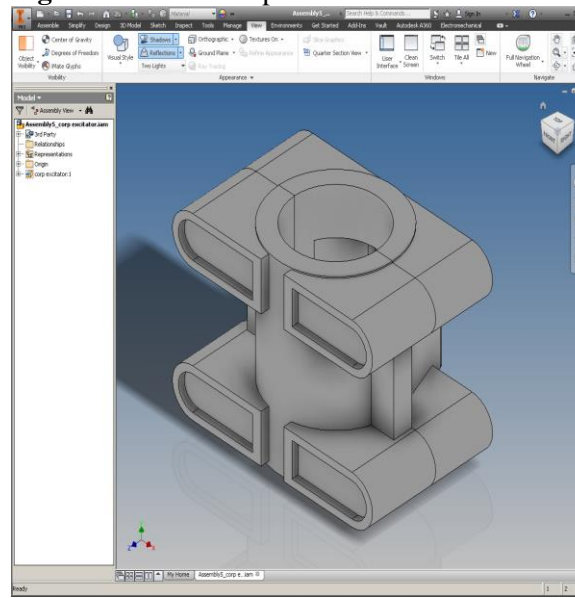

Figure 10. Initial part achieved in Solid Works

Figure 11. Modified part in Solid Works.

Figure 12. The result of the initial part importation in Inventor [9].

Figure 13. Final update in Inventor of the part modified in Solid Works.

5. Conclusions

Based on a detailed and more extended study than what is presented in this paper, the author has reached the following conclusions:

✓ Using CAD design methods (Computer Aided Design) has completely changed the traditional way of working on a board, which implies drawing 2D views from the beginning, both at the level of parts and of ensembles.

✓ Applying the modern aspects of the present day design methods implies the achievement of various informatics models capable to encapsulate the information describing the geometrical

three-dimensional form, the alpha-numerical information which describes the materials the parts are made of and their chemical and physical properties, subsequently allowing the virtual simulation of various work conditions in view of determining the limits of the project or tracking possible mistakes in the design.

✓ The formats conversion method remains the cheapest, the most accessible and widespread method of import/export between design software applications produced by various companies. It must be mentioned that this transfer method by means of conversion can be applied not only between software of the same type (CAD-CAD), but also between software with different objectives (CAD-CAE, CAD-CAM).

✓ The integration method based on *.prt type files, initially a property of UGS and now of Siemens, is a very appealing alternative in the field of assisted design because the *.prt file contains all the information necessary to three-dimensional modelling, but also to the finite element analysis, or fabrication simulation, and to managing the documentation for the whole life cycle of the product. Moreover, it must be stated that the respective file also contains the execution drawings for de designed part or the overall drawing for the total result of the design work.

✓ There is a tendency to launch new technologies for manufacturing/importing/transferring files intended to overcome the flaws of the traditional transfer methods. In this respect, mention must be made of the AnyCAD technology from Autodesk, and of the Unite Technology from PTC.

✓ The essence of AnyCAD technology consists in the fact that its user imports a 3D model as a reference file and creates a link to it instead of converting it as has previously been the case. This allows that any modification achieved at the level of the reference file be automatically transmitted in Inventor.

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