Title:
Integration of Neutral/Re-Imported Models for Assembly Update

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Introduction:

The nature of product development has been changed over the years. Every component required for a product was used to be produced in-house by the factories. Now a vendor-based approach is followed by the factories. The vendors provide required components for the product to be produced. The approach for digital product or the CAD models has also changed due to availability of different specialized solutions. In other words, there are suppliers of CAD models as well. The CAD models are shared in various formats depending on the requirements of the consumer's application or Target CAD System (TCS) but it also depends on the functionality of the supplier's application or Source CAD System (SCS). The required formats can be native such as ".sldprt" which is a SolidWorks file or a standard format. The native format would be the one supported native to the TCS whereas standard formats are those which are now internationally accepted such as "Initial Graphics Exchange Specification (IGES)" or "Standard for the Exchange of Product model data (STEP)" etc. These models are referred to as neutral CAD models which cannot be used as native CAD models because of limitations of the information available in them as well as the restrictions of the TCS for neutral file.

The CAx integration by Jia et al. [3] studied integration for a multibody mechanical system focused on data integration for different CAx systems than for revision of the CAD files. The data integration for different workbenches for a particular file type is the area of interest of commercial CAD vendors. One such application is Dassault Systèmes Spatial 3D InterOp [1]. The study for the revision of the CAD files in the standard formats is highlighted by Kirkwood et al. [6] in which a design change vector technique was proposed for single file whereas assembly constraint solving is an old area of research in which degree of freedom analysis have been used by Kramer [7]. Automatic constraint solving has been studied by Kim et al. [4], [5]. A method for integration of native file formats based on concurrent modeling was presented in [13], but this study [13] reported that components were modeled at required position but constraint transfer was unsuccessful. The problem of integration of motion linkage data for neutral file format like STEP have been studied [8], [9]. The problems in the exchange of neutral file formats such as variations of topology or geometric information has been studied by Dimitrov et al [2]. The persistent naming [12] of the geometric entities would be required to solve geometric constraints. To understand this naming, grasping the knowledge of internal geometric modeling kernel would be necessary [10]. The persistent naming method for STEP files has been provided in the ISO-10303-57 [11] but the implementation of this method would require work from the CAD vendors with a need for conformance of implementation from all CAD systems.

Main Idea:
Suppose there are two CAD systems, a “Source CAD System (SCS)” which provides CAD models and a “Target CAD System (TCS)” which receives CAD models. A model is imported into TCS for some upstream applications like assembly, product manufacturing information (PMI) etc. Then the model is changed in the SCS and is re-imported in the TCS, even if the entities of interest are unchanged, updating the assembly model may lead to a problem case.

The problem here can be described in an analogy to “a group tour to another country”. The members of the group have their identification in their own country which is analogous to the persistent naming within a single CAD system either topology based or geometry based. The group members could be analogous to the entities within a CAD model. When this group requires to go to another country each person in the group would require an identification document i.e. passport which would be analogous to transferring a file in neutral format from an SCS to the TSC. Thus, each entity within the STEP file would be required to have identification information. Moreover, the TCS should be able to use, preserve and update the identification information attached with each entity.

In this paper, an integration problem is discussed with respect to assembly constraint’s reference when components of assembly are available in neutral file format. The proposed methods deal with the integration problem when imported geometry files do not contain the required information for integration with the TCS. This integration/associativity problem arises when an imported geometry model “α” is replaced with a re-imported model “α’”. When this happens, the TCS loses the required reference information which would be otherwise present for the case of the native file. This problem can be presented in Fig. 1.

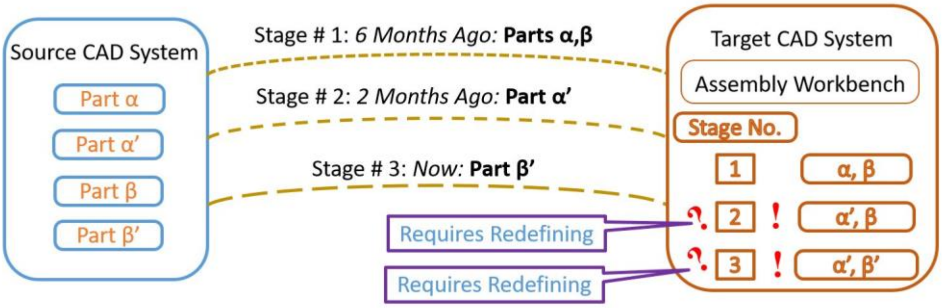

Fig. 1: Unable to solve Assembly for Revisions of α and β.

Commercial CAD systems are important for the industrial applications; therefore a few were checked for this problem. These CAD systems have different geometric modeling kernels and naming rules. This integration problem was emphasized during development of the ISO-10303-57. For the required assembly scenario for re-imported geometry models, the following CAD systems were tested.

- Dassault Systèmes CATIA V5
- Siemens NX 10.0
- Autodesk Inventor 2017
- PTC Creo Parametric 5.0

The term of “Exchangeable Persistent Identifiers (EPiDs)” is referred to functionality of attaching a string to each entity within a step file. This type of attachment in supported by the STEP format whereas mechanism of attachment could be manual to the STEP file by using a text editor or by using a CAD system to export such kind of identifiers. This identification is entity based neither geometry based nor topology based. Therefore, it could be any kind of information. In Tab. 1, “Exchange without EPiDs”, means that no identifiers were attached and exchanged within the STEP file whereas in case of

"Exchange with EPiDs", EPiDs were included in and exchanged within the STEP file from the SCS by exploiting the name attribute for entity in ISO 10303-42. The results of the CAD systems varied depending upon their functionality for handling STEP file. The results obtained from the CAD systems are summarized in Tab. 1. Siemens NX 10.0 showed promising (△) results when finding the required entities for constraint solving based on EPiDs where as EPiDs attachment (△△) was supported only for those entities whose names are manually added. There could be problem when only EPiDs are added for a particular face, and that face and its edge is referenced for two different constraints. The identifiers in the STEP file can be viewed and edited in Siemens NX 10.0 by opening in a single ".prt" file. The problem occurred in this case was the automatic attachment of edge identifier same as the face EPiD if edge is not given an EPiD separately. PTC Creo (△) can automatically name the entities but there was problem in persistent of edge names. Autodesk Inventor's results were partially successful when STEP file was used as imported geometry file whereas there was no issue when STEP file was imported as reference model but Inventor was unable to attach or retrieve EPiDs. The "X" in table 1 is used when the particular CAD system do not provide functionality for inclusion or retrieval of EPiDs.

"Constraint Solution" in Tab. 1, indicates what happened when a component was replaced with an imported geometry component. "Holds-Different Association" means that there was an error in finding the required name or inferred information was associated with a different entity. "Requires Re-defining" refers to the user's choice of redefining the constraint whereas "Goes to Assembly constraint Environment" means that the user has to select the reference for the imported geometry file before going any further. "Shows error" means that an error exception was thrown by the TCS.

Commercial CAD Systems	***Function "Replace Component"***	***Exchange without EPiDs***	***Exchange with EPiDs***		
		Constraint Solution	EPiDs *inclusion in STEP file*	EPiDs *Retrieval in Assembly*	*Constraint Solution*
CATIA V5	O	Holds-Different Association	X	X	Holds-Different Association
NX 10.0	O	Requires Re-defining	△△	△	Solves Constraints/may cause problem
Creo Parametric	O	Goes to Assembly Constraint Environment	△	X	Goes to Assembly Constraint Environment
Inventor 2017	O	Shows error	X	X	Can Show error

Tab. 1: Results of Tested CAD Systems for Assembly Update.

The proposed system to address this problem, shown in Fig. 2, consists of two methods: one (I) is the ideal method, in which EPiDs should be added in the SCS and used in TCS. An assembly model should be defined in the TCS. In this method, the SCS and TCS should have functions to add and use EPiDs to find entities respectively. The SCS should be able to export the identifiers allotted to the entities in the STEP file.

In the second method (II), work should be done in the TCS. First an assembly model of the required product should be defined in the TCS. The components should then be replaced by re-imported version of the same component. The constraints and constrained geometry information would be saved in the integration program, this information will be used for the vector comparison to find the required reference information for re-imported file. The constraints should then be applied to update assembly.

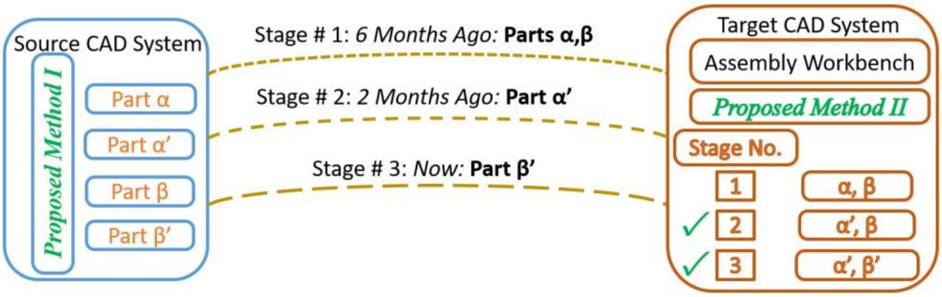

Fig. 2: Solves Assembly for Revisions of α and β.

The proposed system's implementation was performed using PTC Creo Parametric 5.0 as SCS and Siemens NX 10.0 as TCS. For "Method I", EPiDs were interactively added and updated in the SCS. These files were imported in the TCS for the assembly constraint solution. The persistence for EPiDs was checked before exporting as STEP file.

In the case of "Method II", there were no EPiDs and assembly constraints were analyzed in an integration program based on the NXOpen API. The constrained geometry information from the old model was used to create a set of unit vectors which was used to find the reference geometry in the new revised model. After finding the required information constraints solving was performed by the application automatically. This approach is limited to small changes and minor topological modifications. The method used here was similar to Siemens history free approach. The implementation environment was Visual Studio 2017, 64-bit and Windows 10 Professional 64-bit.

Conclusion:

The proposed system proves to be fulfilling the requirements of the integration for solving the 3D geometric constraints for assembly modeling and user's requirement of low-level entity selection was removed. The technique described in the proposed method "I" is robust for the TCS whereas it requires extensive work in the SCS. This is because of interactive EPiDs inclusion and update in the SCS. The problem with this approach is the requirement of effort from the suppliers of the neutral model and from the consumer in their particular CAD systems if functionality is not already available. The method described based on vector comparison would be more computation expensive and may be unable to support for the simultaneous geometry and topology changes. This problem may be solved by following a procedural method in which first geometry is changed and then topology. Therefore, functions that the proposed method "I" can perform in a simple file replacement, proposed method "II" would require two replacements to achieve the required assembly model. In a nutshell, the proposed integration methods are robust and would require little to no input from the user of the TCS thus reducing time-consuming tasks. For future work would be to make EPiDs independent of the CAD system and their persistence would be check automatically.

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