Assembly Solving for Neutral Re-Imported Product Models

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Abstract. Computer-Aided Design (CAD) product data integration is essential for applications in Computer Aided Engineering /Manufacturing (CAE/CAM). The data integration problem becomes acute when the geometric modeling work is outsourced for a project. These geometries will then be imported for the project. The product design is based on a recursive method which requires that several revised imported geometry models have to be re-imported into the Target CAD System (TCS). These re-imported models can be difficult to handle, particularly for Original Equipment Manufacturer (OEM) companies. The iterative nature of the product design requires that several operations be performed on the re-imported files, even if the first import has been dealt with. In this article, the author discusses two methodologies for integrating the data of re-imported models. The proposed approaches involve the use of exchangeable persistent identifiers for the imported geometry file, which have to be included and shared for every revision of a particular imported geometry file. This is in contrast to history free vector comparisons based on entity vertices for finding the required reference information of a particular constraint. The proposed methods may reduce the lead time that is wasted on fixing the re-imported geometries.

Keywords: Product data integration, 3D geometric constraints, Persistent naming, Neutral files, STEP, Imported geometry, Assembly update
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1 INTRODUCTION

The process of product development has evolved over the years, from factories producing everything in-house to a vendor-based product approach. This evolution has led to increased employment opportunities, and the sharing of risk factors. Advances in Computer-Aided Design (CAD) technology have also led to a great expansion of available products. Design trends in industry have also changed with the introduction of different specialized Computer-Aided Engineering (CAE) solutions. As a result, not only producers and suppliers, but consumers increasingly utilize digital products. Most CAD models can be exchanged depending on consumer's requirements, but producers have some limitations. The requirements depend on the respective CAD systems used by the producers as the Source CAD System (SCS), as well as the consumer or Target CAD System (TCS). The file format requirements may vary because of the heterogeneity of CAD systems and the vast number of applications. To address this diversity, standard formats have been defined, such as Initial Graphics Exchange Specification (IGES), Standard for the Exchange of Product Model Data (STEP) etc. These standard formats are often called neutral models, although persons using TCS may call them imported geometry models.

Using these imported geometry files can be more limiting compared to using native file formats, which are produced for the respective commercial CAD systems. The imported geometry files often require extra work due to these limitations. To reduce both lead times and non-productive tasks, while also streamlining the product development process, integrating CAD model data is essential. Integration [18] refers to the process of integrating information for CAE applications such as assembly design, product manufacturing information and finite element analysis, etc. Integrating CAE data between systems is considered a highly important field of study, however, integrating revised versions of a model for downstream applications requires further format-based research.

In this integration study, the focus is on revised versions of the model, with the files in neutral format. The integration functions available in commercial CAE solutions only provide integration support for native file formats, and this requires tedious repetitive tasks for standard files. This type of integration should work in a seamless manner for all types of CAD models. Current CAD model integration is format-based, with limited research in the area of neutral model integration [18]. Imported geometry models are widely used but require tedious non-progressive tasks because they have no standard integration for different solutions. The ease of working with imported geometries could be improved if their integration process was simple and effective. Work is currently being done to solve this integration problem. This integration problem of neutral CAD files is shown in schematic form in Figure 1.

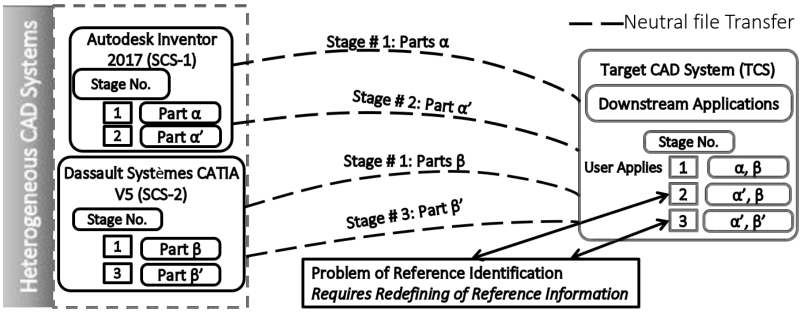


Figure 1: Problem of neutral re-imported models in the TCS for downstream applications

Whenever a TCS works with an imported geometry file from a SCS for downstream applications such as assembly constraint solving, Product Manufacturing Information (PMI), Numerical Control (NC) manufacturing, etc., the TCS defines references to entities of the imported geometry file. The required file is then edited in SCS and re-imported into the TCS. When the imported geometry file is replaced with a newer version of the same model, the information that was assigned to the imported geometry when it was first imported should work with the latest import: but this does not happen. The required information is lost because the TCS cannot find the required reference information. As shown in Figure 1, that problem exists for stages two and three. Figure 1 explains the integration problem for neutral files. In the first stage, six months ago, parts α and β were designed in SCS and sent to the TCS for downstream applications. In the downstream applications workbench of the TCS a user interactively applied functions to the first stage. Stage two came two months ago, when α was to be replaced with α' in the TCS. At this stage, the TCS was unable to find the required reference information which is essential to retaining the downstream functions. This made it necessary to have user interaction to obtain the reference information.

Assembly constraints solving can be considered the most important stage in product development, therefore, in this article, the problem of assembly constraint solving is discussed. This paper has been divided into eight sections. Section 1 provides an introduction, while section 2 deals with related works. Section 3 explains the integration problem in commercial CAD systems and section 4 explains possible reasons for the problem. In section 5, a solution is proposed for assembly constraint solving in the re-imported geometry problem. Further, section 6 explains possible implementation scenarios and their results. Finally, sections 7 and 8 provide the limitations and future works, respectively.

2 RELATED WORKS

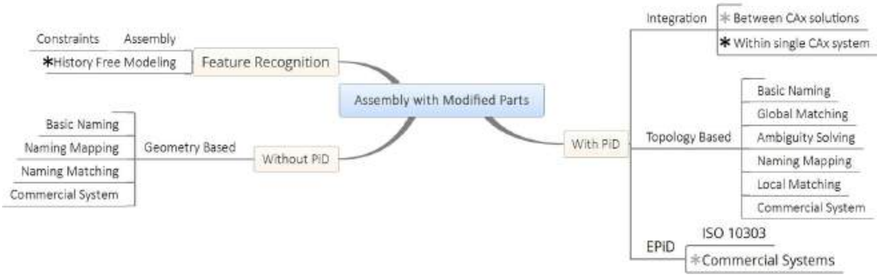


Figure 2: Technology tree of related research

Figure 2 shows the technology tree for this research. The issue of CAD data exchange and its problems are quite old. A tool was developed to check the conformance of CAD data conversion, verification and healing [20]. The integration issue of packages was explained by Piegl [25]. There are studies which deal with Computer Aided Systems (CAx) integration. The CAx integration by Jia et al. [13] studied integration for a multi-body mechanical system with a focus on analysis data integration for different CAx. In that study, a user interactively selects the low-level entities to apply constraints to the multi-body assembly, while integration is performed for different analysis tools. A method of integrating mechanism information for neutral files such as STEP was performed in [21, 22]. In this method, STEP files were used for integration with Siemens NX. This method defines link information from the Siemens NX to be exported with a STEP file to implement ISO 10303-105 [11]. Moreover, there is a standard for persistent naming in ISO 10303-57 [12] named "Integrated generic resource: Persistent identification of elements in procedural shape modelling".

Three dimensional (3D) geometric constraints are different than linkage information. Data integration of different work benches for a particular file type is the area of interest for commercial vendors. One such application is Dassault Systèmes 3D InterOp [1]. The file format employed in 3D InterOp's persistent identification was from SolidWorks, being from the same vendor. As a result, the functions of the same geometric modeling kernel may have been exploited in this method. However, based on the video [1], the identifiers were different than SolidWorks naming [6]. A study for revising CAD files in standard formats was reported by Kirkwood et al. [18]. In that study, a design change vector technique was proposed to investigate the changes in the model. This study was implemented in an IGES format with only a change in one length. Their proposed system is a standalone system which works ad-hoc for applications. However, it was not shown how these entities are referenced, except for their import of IGES and export as a Parasolid file. A Virtual Persistent

Identifier (VPI) technique was implemented in [18]. However, how VPI [17] can be implemented for different TCSs is not known, except for its application in Finite Element Analysis (FEA) with an unknown FEA solver.

In general, models that are used for FEA analysis are simplified with respect to their level of detail. This work was performed for a single CAD model only. Product development not only requires FEA but assembly constraint solving. 3D geometric constraints solving could be regarded as the most important stage in the product development phase. Because of the current trend of using parts interchangeably, and module-based products, assembly solving will be required for finite element applications as well. Assembly constraint solving is an old area of research [30]. Degree of freedom analysis has been used by Kramer [19]. Application specific algorithms have been developed for assembly solving [31]. Automatic constraint solving has been studied [15, 16, 23, 28]. The topic of assembly sequence planning [32] is outside the scope of this research. These methods are independent of the persistent names of the models.

The problem under consideration is integration for re-imported geometries for assembly constraints solving. A method of collaborative procedural modeling was proposed in [3] which used a server to control the modeling information, but no information was provided about assembly. In a method [27] for integrating native file formats based on concurrent modeling, the components were modeled at the required position but constraint transfer was unsuccessful for assemblies. The problem of integrating motion linkage data for neutral file formats like STEP have been studied [21, 22]. This study performed the integration for the transfer of linkage information and motion analysis, but not for constraints. Therefore, in downstream applications of CAD models, assembly constraint solving is the most recent topic, followed by linkage definition and motion simulation.

In the present research, importance is given to a single system solution for integration. A vector comparison method based on a history free approach is introduced in Figure 2, and indicated with a black asterisk. Single system integration avoids the pain of linking different systems, and requires no input from a user. In addition, while a combination of currently available commercial CAD systems can be used by naming each entity manually, this required an intermediate system, indicated by the gray asterisk in Figure 2. This combination comes under EPiD and is used for integration between CAx solutions.

3 NEUTRAL FILES PROBLEMS IN COMMERCIAL CAD SYSTEMS

Commercial CAD systems are important in industrial applications, therefore a few of the commercial CAD systems were checked for this problem. These CAD systems have different geometric modeling kernels and persistent naming rules. These CAD systems have specialized customer circles depending on the field of application, since CAD tools are now specialized for specified purposes. Therefore, the supplier of the CAD models may use one CAD system with licensed support and ease of use for modeling such as SCS, whereas the TCS could use one which supports downstream applications at the Original Equipment Manufacturer (OEM), company such as NC manufacturing. Ultimately, both the systems have equal importance. For the required assembly scenario for re-imported geometry models, the following CAD systems were tested for the assembly constraints problem with the re-imported geometries.

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

The neutral file format depends on the lowest of the supports provided by the TCS and the SCS. There is a method of attaching a string to an entity within the STEP file which is defined as the "name" attribute of the entity "representation item" in ISO 10303-41, which will here be designated exchangeable persistent identifiers (EPiDs). The EPiDs remain depending on the presence of the entity in the CAD model. For example, if an EPiD is attached with a face entity, which is later edited to have a round boundary, the EPiD remains there, whereas, if such an identifier is attached to an edge entity which was later changed to round, then such an entity would lose the identifier. The change will be supported if and only if the CAD system in which the model is being modified supports the EPiDs update based on the old EPiDs. Moreover, the EPiDs inclusion functionality is limited even for single export. This type of functionality is not common within the CAD systems when a modified neutral file is imported. Therefore, when neutral files have to be exchanged with an entity name, where the entity could be a face or an edge, the critical functionality which SCS must have is to able to name the entities as well as export those names within the STEP file. TCS must be able to use those identifiers. The method of EPiDs attachment can be tedious, which will be explained later. The systems which allow export of EPiDs within the STEP file are:

- Siemens NX 10.0 (manual attachment)
- PTC Creo Parametric 5.0 (manual/automatic attachment)

The other CAD systems Dassault Systèmes CATIA V5 and Autodesk Inventor 2017, lack the naming functionality. For all tests the SCS used was PTC Creo Parametric 5.0 Student Edition. The number symbol "#" in the part name indicates different revisions of the same component. The components consisted of "body(stationary component)" and "rot(rotating component)" where "rot" contains a hole in which the circular extrusion of the "body" can be set. This version allows the interactive addition of EPiDs to the CAD model, and those identifiers can be exported as STEP AP-214. Creo has functionality for STEP AP242 export. The details of the assembly's components' replacement tests for each CAD system are explained in the following portion.

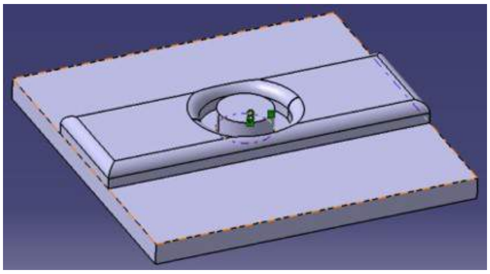
3.1 Dassault Systèmes CATIA V5

In case of CATIA V5, the first base assembly model was made, shown in Figure 3 (a), based on those STEP files which contained the identifiers of the entities which might be required for constraint solving. The base model was named "body-#" with different variations. The other component was the rotating component model named "rot-#". The constraint solving was successful as long the topology of the base model was not changed, i.e., if it was not replaced with a component which contained a varied topology, where the topology of the rotating component might vary as well as the geometry of the base component. Figure 3 (b) shows that the geometry of the "body-#" component was edited. The problem case appears when the "body" component is replaced with a varied topology "body-#" component Figure 3 (c). This problem does not depend on the variation of the "rot-#" component, as shown in Figure 3 (d). In other words, if topology of the reference entity is changed upon component replacement, there is problem in assembly constraint solving.

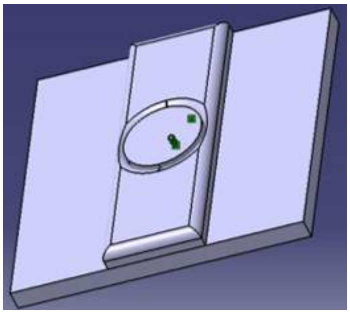
3.2 Siemens NX 10.0

Siemens NX 10.0 delivered the most promising results in this test scenario, but the problem was present when there was no addition of EPiDs within the file, as shown in Figure 4 (b). In the second test, Figure 5, only face named components were used, and "Touch" and concentric constraints were defined. When replace component function was used, there was an error, because the same name was being used to call a surface as well as an edge in Figure 5 (b).

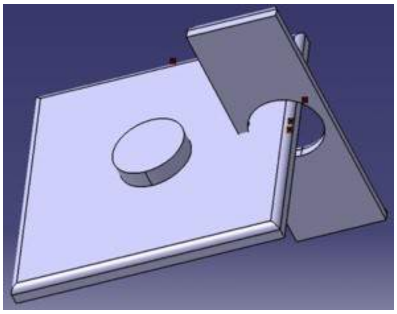
For the third and fourth tests, now the edges were given EPiDs as well as faces which were unique within a single part. For the fourth case, "Touch" was used as face based constraint and "Concentric" was used as edge-based constraint. This time different combinations were tested, as shown in Figure 6 (a) and (b), but the constraints remained valid. Different variations of constraints can be used to make the same visual assembly in Figure 4 (a) and Figure 6 (a), therefore, to conduct the fourth test, two "Touch" constraints were used, one of which was face based and the other was edge based. The variations in the components used were the same as those which were used for the third test. It can be seen from Figure 7 (b) that the touch constraint



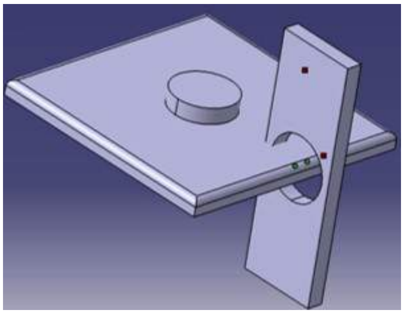
(a) rot-2 and body-0



(b) rot-2 and body-1

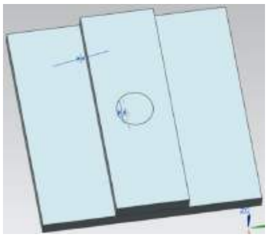


(c) rot-2 and body-2

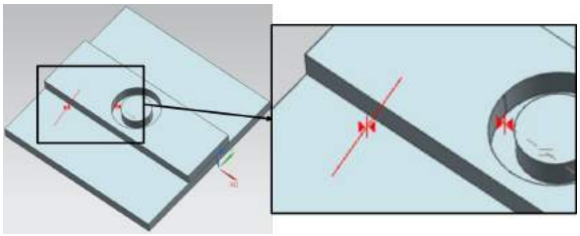


(d) rot-1 and body-2

Figure 3: Dassault Systèmes CATIA V5 test results with (c) and (d) problem cases



(a) rot-0 and body-0



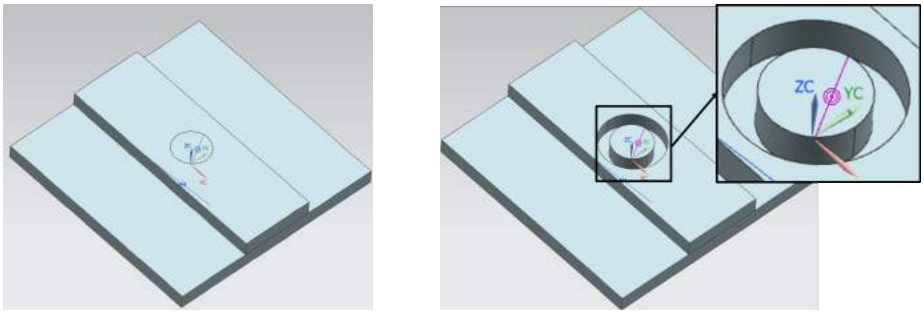
(b) rot-1 and body-0

Figure 4: Siemens NX 10.0 first test (no EPiDs) result with (b) Problem case

remained valid. In Figure 6 (b), it is evident that the edge-based concentric constraint remained valid for varied geometry.

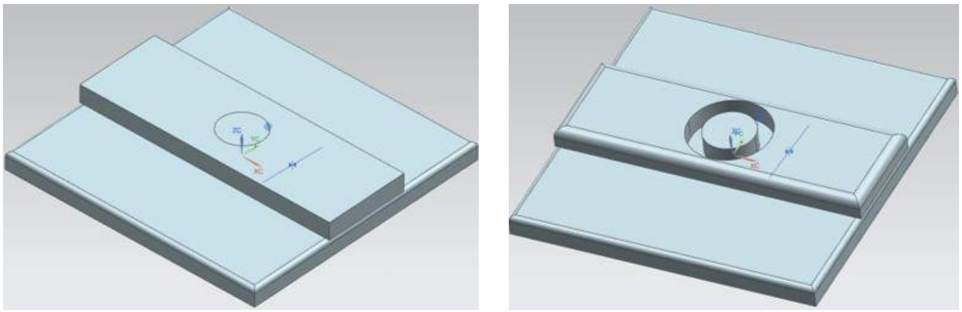
3.3 PTC Creo Parametric 5.0

For the PTC Creo Parametric assembly, two coincident constraints were used. When the component replacement function was used, "the assembly constraint reference user input" window appeared asking for the user input for constraint. Therefore, the user did not have to open the assembly constraints environment. The assembly constraint environment was automatically opened requiring just a mere click from the user as shown in Figure 8 (b), whereas as Figure 8 (a) shows the original assembly model.



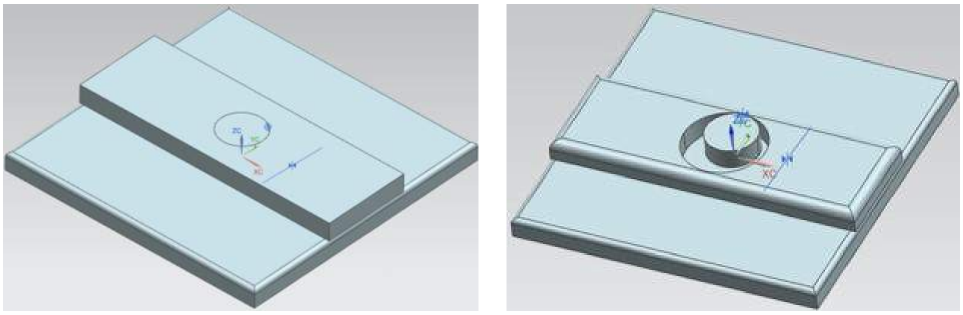
(a) rot-0 and body-0 (b) rot-1 and body-0

Figure 5: Siemens NX 10.0 second test (only face-based EPiDs) result with (b) Problem case



(a) rot-0 and body-2 (b) rot-2 and body-2

Figure 6: Siemens NX 10.0 third test results

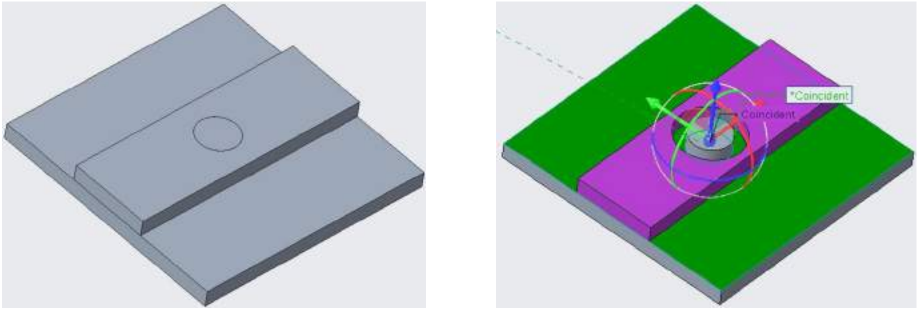


(a) rot-0 and body-2 (b) rot-2 and body-2

Figure 7: Siemens NX 10.0 fourth test results

3.4 Autodesk Inventor 2017

Autodesk Inventor 2017 can import files in two ways; one is the reference model and the other is the converted geometry model. The 'Reference Model' option maintains a link to the selected file which enables you to monitor and update as the model changes. This option is used if the design is evolving, and you are not required to edit the referenced model. The 'Convert Model' option creates new inventor files which are not



(a) rot-0 and body-0 (b) rot-1 and body-0

Figure 8: PTC Creo Parametric 5.0 test result with (b) Problem case

linked to the original. This option is used if you plan to reuse and modify the model for a new design. Since OEMs require operations to be performed after import, the converted model was studied. Two constraints, "Mate" and "Insert", were used to make the same constrained assembly. When a component was replaced, the constraints remained intact for all variations of the revised imported geometry files for the "rot-#" component, and only for the geometric changes in the "body-#" component. When "body-#" was replaced with a revision which contained a variation in topology, there was a problem as shown in Figure 9 (e).

4 REASONS OF INTEGRATION PROBLEM

There can be many reasons for integration problems for neutral imported geometry files. The possible reasons for integration problems with neutral CAD files can be divided into the following categories:

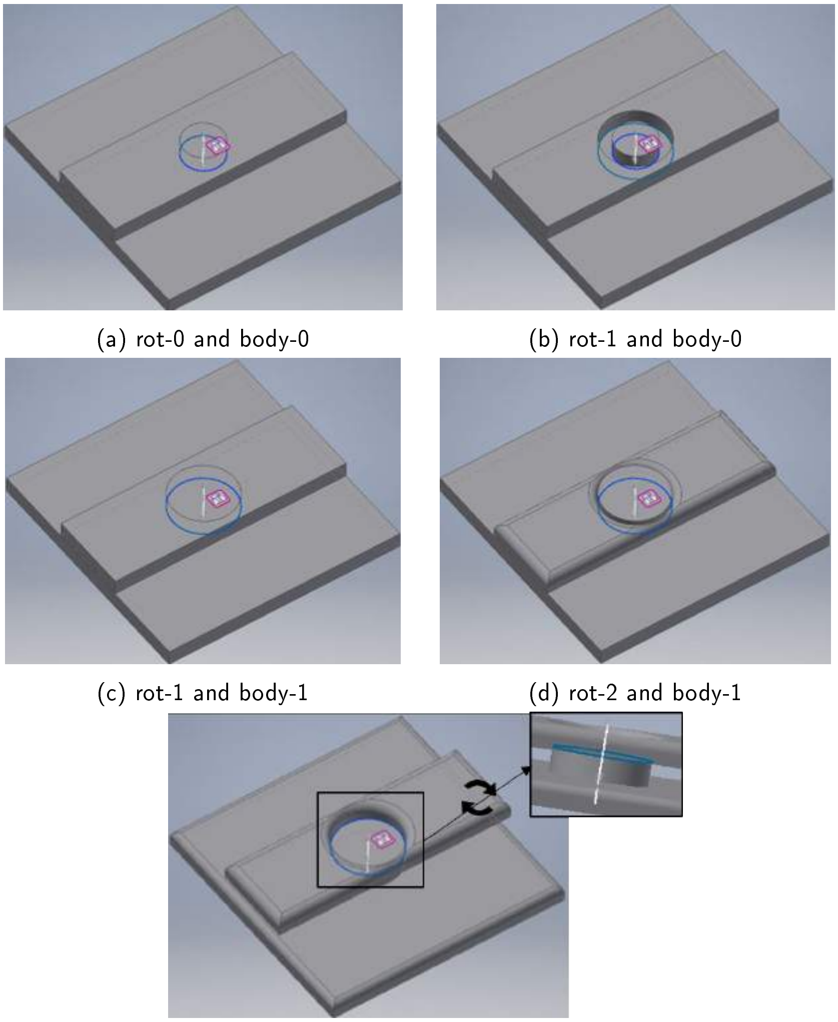
- Precedence of the native file format
- Heterogeneous geometric modeling kernels
- Persistent Identification for standard files

4.1 Precedence of the native file format

Commercial CAx are based on one particular geometric modeler which acts as the source of integration for all applications for all solutions from one vendor. This can be called the associativity of the CAD file for CAx. The small recursive problems caused by neutral formats encourage customers to use the native file for analysis, as compared to neutral formats. Accordingly, there have been studies on developing effective parametric translators [4, 14].

4.2 Heterogeneous geometric modeling kernels

The geometric modeling kernel is like the engine of a CAD system [26], and different geometric modeling kernels have different modeling capabilities. It is quite possible for a number of CAD applications to have the same geometric modeling kernel from same vendor, while CAD applications coming from different vendors would have different geometric modeling kernels. As a result, their way of handling the CAD model would be different. For example, if a circle is extruded, while one CAD system creates one circular face Figure 10 (a) the other could divide it into two faces, as in Figure 10 (b). For exchange in neutral format, the required model must be translated by the source geometric modeling kernel, and before use it must be translated again



(a) rot-0 and body-0
(b) rot-1 and body-0
(c) rot-1 and body-1
(d) rot-2 and body-1
(e) rot-2 and body-2

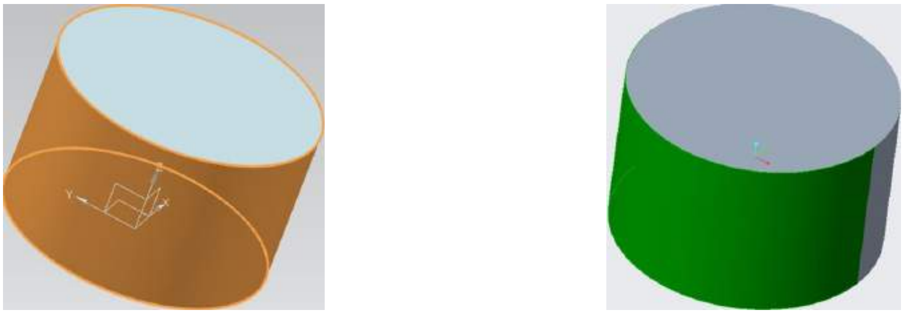
Figure 9: Autodesk Inventor 2017 test results with (e) Problem case

by the target geometric modeling kernel. Therefore, this type of translation limits the integration because of some information missing.

4.3 Persistent identification for standard files

This problem can be described under the umbrella of persistent identification and CAD data exchange. Three definitions should be addressed before starting the topic of persistent identification.

- Persistent naming
- Name matching
- Name mapping



(a) Siemens NX 10.0: One cylindrical surface (b) PTC Creo Parametric 5.0: Two cylindrical surfaces

Figure 10: Different number of faces for similar model requirements

Persistent naming [6] is the basic naming of the entities of the geometric model being constructed within a CAD system that are related to a particular geometric modeling kernel which may later be used for calling that particular entity or referencing that entity for constraints, boundary conditions, and so on. This can be called basic naming and it can be classified into two types: geometry-based and topology-based. The topology-based naming is simple [7] but it may cause a problem of ambiguity. Due to effective research in the area of persistent naming this problem has been solved.

Name matching comes into use when this ambiguity problem needs to be solved. Different algorithms have been defined to solve this ambiguity problem [24]. All these depend on working within the same system on a particular CAD model. For the parametric exchange of CAD models, different name mapping techniques have been defined to check the conformance of the exchanged model, but this requires understanding the rules and adding the names of entities of the internal geometric model (IGM) [24]. All these names are entity dependent, therefore these names will change every time the entities are changed.

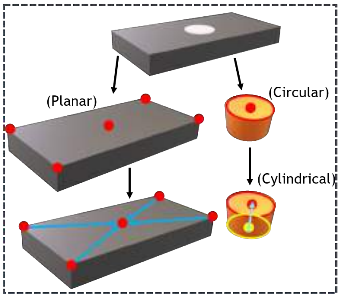
A persistent naming method for STEP files has been given by ISO 10303-57 [12] but it only covers the persistent naming of procedural models. A mechanism for entity names of revised neutral file is not available. Persistent naming for modeling applications is effective, such as when an entity is completely lost. Such a problem occurs when an edge is split into two and fillet is performed on one of the edges, meaning a portion of the edge remains but the other one is gone. The Quality Information Framework [33] (QIF) provides QIF Persistent Identifiers (QPID) which are universally unique identifiers for metrology applications for a single model, with no information provided for the revision of the models. Research on persistent naming has been performed by Hoffmann [10] but there are other techniques like geometry-based semantic ID [29] for feature based parametric models and constraint-based naming [2].

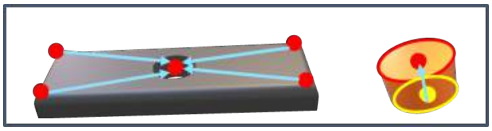
5 PROPOSED METHOD: HISTORY-FREE APPROACH

The proposed method considers the component and constraint names. Furthermore, the information required for integration considers the type of entities which constitute the constraint, and whether they are face-to-face, edge-to-edge or face-to-edge. Since the constraints are originally based on an older version of the model, the mapping technique should consider the older version, to re-map the constraints' references for the newer version of the model. Therefore, to compare different versions of the same component, face or edge information is extracted from the older version of the component and is compared to the newer version of the component. There can be two types of comparisons. One would be the face-based comparison and other the one would be the edge-based comparison.

5.1 Face-Based Comparison

If the constraint geometry is a face, then a face-based comparison is used to find the corresponding entity in the newer version of the file. In this case, the topological face type is saved in the data base, based on the old version of the component present in the assembly. The database also saves the number of edges that belong to the face. If the face contains a circular edge, such as in Figure 11, then unit vectors are calculated to one of the vertices of the circular edge from all the corresponding vertices. Then this information is saved. For a planar face containing a circular edge, which makes a circular face, the unit vector that explains the direction of the circular face would be given precedence over other vectors. Such a case can occur when there is no circular edge present in the face, and it is only a planar face, as in Figure 12. Then, the unit vectors are defined based on the faces adjacent to the face of interest. Same operations are performed on the revised imported geometry file and a comparison is performed to find the required entity of interest for constraint solving.

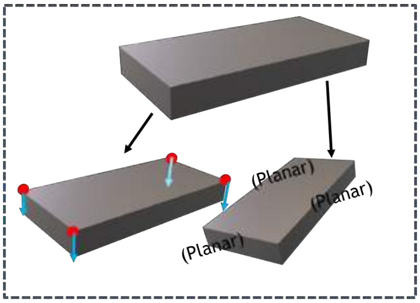


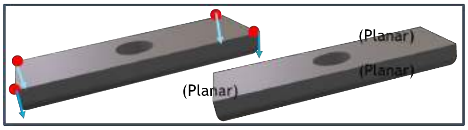


(b) Matching vector set in the edited model

(a) Vector set of the constrained geometry in the old model

Figure 11: Matching face updated with fillets on its edges





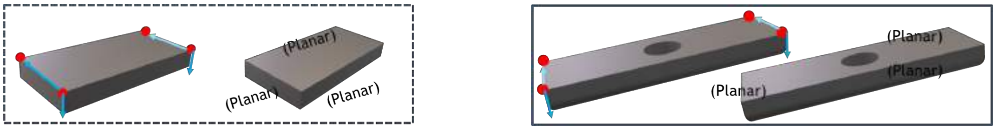
(b) Matching vector set in the edited model

(a) Vector set of the constrained geometry in the old model

Figure 12: From just a planar face only to the hole containing face

5.2 Edge-Based Comparison

If the constraint geometry is an edge, then an edge-based comparison is used to find the corresponding entity in the newer version of the file. The stored edge type may be linear, circular or elliptical etc. The corresponding faces of the edge of the interest are used to calculate the unit vectors. If the edge is circular, then the unit vectors are calculated in the same way as shown in the Figure 11. The difference would be that now the program will search for a circular edge rather than a face. There could be a case with a linear edge. In this case, the unit vectors are defined based on the edges which correspond to the edge of interest, and have one vertex in common with the edge of interest, as shown in Figure 13.



(a) Vector set of the constrained geometry in the old model (b) Matching vector set in the edited model

Figure 13: Matching linear edge

6 IMPLEMENTATION

6.1 Selection of SCS and TCS

The implementation of the proposed system was divided into three sections. Since a number of CAD systems were used during the testing phase, the CAD systems used for implementation were selected from the previously tested CAD systems. Siemens NX 10.0 was selected as TCS, and the SCSs could be Siemens NX 10.0, Autodesk Inventor 2017, PTC Creo Parametric 5.0 and Dassault Systèmes CATIA V5. Siemens NX 10.0 was used as TCS because of its open application programming interface (API), NXOpen.

The PTC Creo Parametric 5.0 student edition was used as the intermediate system because of its functions that allow EPiDs to be added within the STEP file. Autodesk Inventor was used because it was mentioned in the original problem scenario. The EPiDs were added by selecting entities. Those EPiDs could be exported in STEP as AP-214 files. Siemens NX 10.0 has functions to call the exchangeable attached identifiers as well as provide functions to set entity names using the TCS after translation, which can also be used for entity selection as well as solving 3D geometric constraints.

6.2 Manual EPiDs inclusion before TCS

In this case, two SCSs were used which do not allow the inclusion of EPiDs in the STEP file. PTC Creo Parametric 5.0 was used as the intermediate system where a user interactively added EPiDs, as shown in Figure 14. The record of the EPiDs was maintained and updated for every revision of the component. Therefore, a single STEP had to go through the import and export process twice before importing into the TCS. The method of EPiDs attachment was tedious because the user has to select each entity and add information while maintaining a record the EPiDs. In this kind of system, the EPiDs of interest may be attached, leaving the rest empty, but this can lead to problems.

The TCS selected was Siemens NX 10.0 because of its ability to read the information attached within the STEP file. First, an assembly model was created and saved in the TCS with the required components. The assembly contained two parts, "body-#" and "rot-#", with different variations, such as "body-0 to 2" and "rot-0 to 2". The saved assembly model was then updated for the revised components. When the component replacement function was used, the constraints were based on the same EPiDs in the re-imported geometry file as those which were present in the previous version. In this of type integration the persistent naming for entity selection in the TCS may be different, but as long as the EPiDs remain the same, the required entity can be referenced for assembly applications. One can find the implementation results at [5].

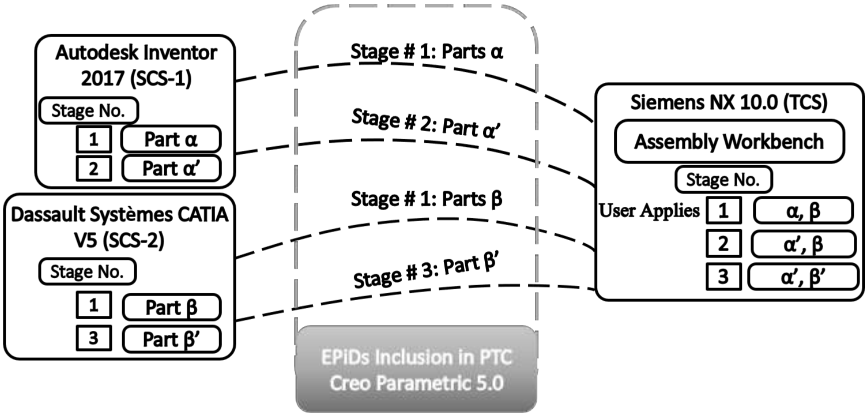


Figure 14: Implementation with EPiDs

6.3 History free vector comparison

In this case there could by any SCS, but Dassault Systèmes CATIA V5 was used as the SCS. There was no intermediate system for the inclusion of EPiDs. A similar process was followed, making an original assembly first and then updating the components. The proposed system works under the TCS as shown in Figure 15. The assembly model used for this case also consisted of the same parts, as explained in the former section.

Since EPiDs were not present, this leads to two cases; one where the geometry was changed while keeping topology constant, and other where topology was changed keeping the constraint geometry constant. In such cases, the system tries to find entity names for a particular constraint by analyzing geometry or topology based on face/edge-based vector comparison. Then the persistent name of the required entity was used to apply the constraints. One can find the implementation results at [8, 9]. For this case there was no need to open a file, the user could only select the component replacement through the application. The TCS used was Siemens NX 10.0 because of the NXOpen API, so no separate license was required for this implementation.

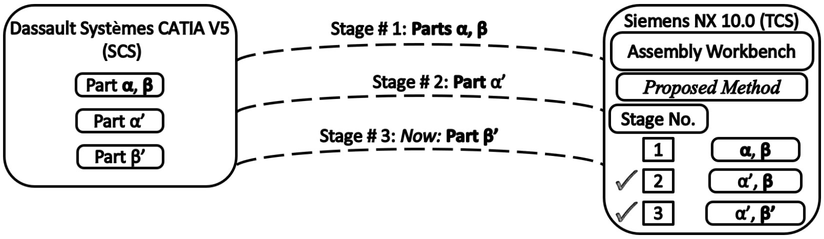


Figure 15: Implementation without EPiDs within TCS

7 LIMITATIONS

The results of the study confirmed that the proposed systems fulfilled the requirements of integration for solving the 3D geometric constraints, and the need for a TCS user for low level entity selection was removed. The technique described in section 6.2 is robust as compared to that described in section 6.3. This is due to the presence of EPiDs, which were added by the intermediate system.

The problem with this approach is that an intermediate CAD system was used for EPiDs inclusion, otherwise this inclusion needs to be provided by the respective CAD vendor. The latter method would be more computation-expensive, and would require user input to inspect for geometry or topology changes. This problem can be solved by following a procedural method, in which the geometry is changed first, and then the topology. The proposed method would not require user intervention at the intermediate stage for EPiDs inclusion.

In a nutshell, the proposed integration methods are robust and would require very little to no input from the user of the TCS. This significantly reduces the time-consuming tasks for user input.

8 FUTURE WORK

The proposed method needs to be matured for the seamless integration of not only 3D geometric constraints but other applications. The EPiDs method could be generally applied if all the CAD systems provided updates and remapping for downstream applications after import in the TCS. The EPiDs entity naming should be automated in the SCS, TCS or in a separate system in such a way that the EPiDs remain consistent after modifications. A vector comparison system could also be integrated for any exception where there was no EPiDs information present for the required entities. Assembly is only one of the applications. The proposed system should be implemented for other applications to increase integration for neutral files, and reduce product development costs.

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REFERENCES

[1] 3D interop's persistent naming translation, accessed on: March 13, 2019. http://blog.spatial.com/persistent-naming-translation.

[2] Capoyleas, V.; Chen, X.; Hoffmann, C.M.: Generic naming in generative, constraint-based design. Computer-Aided Design, 28(1), 17–26, 1996. http://doi.org/10.1016/0010-4485(95)00014-3.

[3] Chen, J.; Ma, Y.; Wang, C.; Au, C.: Collaborative design environment with multiple CAD systems. Computer-Aided Design and Applications, 2(1-4), 367–376, 2005.

[4] Elber, G.; Patrikalakis, N.; Brunet, P.; et al.: Integrated feature-based and geometric CAD data exchange.

[5] EPiD implementation. https://youtu.be/k4F8xBfgNCM.

[6] Farjana, S.H.; Han, S.: Mechanisms of persistent identification of topological entities in CAD systems: A review. Alexandria engineering journal, 2018. http://doi.org/10.1016/j.aej.2018.01.007.

[7] Gao, X.Y.; Zhang, C.X.; Ren, M.Y.; Gao, S.M.; Liu, Y.: A new mechanism of naming topological entities for semantic feature operations. Journal of Software, 7(3), 705–711, 2012. http://doi.org/10.4304/jsw.7.3.705-711.

[8] History-free approach for modified geometry. https://youtu.be/fA-LR6cWh4w.

[9] History-free approach for modified topology. https://youtu.be/KFrB3wMMzg0,https://youtu.be/plgSrM1WvBw.

[10] Hoffmann, C.M.; Juan, R.: EREP an editable high-level representation for geometric design and analysis. Technical Report CSD-TR-92-55, CAPO Report CER-92-24, 1992.

[11] Integrated application resource: Kinematics, ISO 10303-105, accessed on: March 13, 2019. https://www.iso.org/standard/64294.html.

[12] Integrated generic resource: Persistent identification of elements in procedural shape modelling, ISO 10303-57, accessed on: March 13, 2019. https://www.iso.org/standard/66348.html.

[13] Jia, M.; Ding, G.; Qin, S.; Li, R.; He, Y.: Research of design and analysis integrated information modeling framework for multibody mechanical system: with its application in the LHD design. The International Journal of Advanced Manufacturing Technology, 66(9-12), 2107–2122, 2013. http://doi.org/10.1016/j.cad.2004.08.012.

[14] Kim, B.; Han, S.: Integration of history-based parametric translators using the automation APIs. International Journal of Product Lifecycle Management, 2(1), 18–29, 2007. http://doi.org/10.1504/IJPLM.2007.012872.

[15] Kim, J.; Kim, K.; Choi, K.; Lee, J.: Solving 3D geometric constraints for assembly modelling. The International Journal of Advanced Manufacturing Technology, 16(11), 843–849, 2000. http://doi.org/10.1007/s001700070.

[16] Kim, J.; Kim, K.; Lee, J.Y.; Jung, H.: Solving 3D geometric constraints for closed-loop assemblies. The International Journal of Advanced Manufacturing Technology, 23(9-10), 755–761, 2004. http://doi.org/10.1007/s00170-003-1686-y.

[17] Kirkwood, R.: CAD Integration through virtual persistent identifiers and design change requirements, 2018. US Patent App. 15/456,475.

[18] Kirkwood, R.; Sherwood, J.A.: Sustained CAD/CAE integration: integrating with successive versions of STEP or IGES files. Engineering with Computers, 34(1), 1–13, 2018. http://doi.org/10.1007/s00366-017-0516-z.

[19] Kramer, G.A.: Using degrees of freedom analysis to solve geometric constraint systems. In Proceedings of the first ACM symposium on Solid modeling foundations and CAD/CAM applications, 371–378. ACM, 1991. http://doi.org/10.1145/112515.112566.

[20] Krause, F.L.; Stiel, C.; Lüddemann, J.: Processing of CAD data-conversion, verification and repair. In Proceedings of the fourth ACM symposium on Solid modeling and applications, 248–254. ACM, 1997.

[21] Li, Y.: Implementation and evaluation of kinematic mechanism modeling based on ISO 10303 STEP,KTH Industrial Engineering and Management, 2012.

[22] Li, Y.; Hedlind, M.; Kjellberg, T.; Sivard, G.: System integration for kinematic data exchange. International Journal of Computer Integrated Manufacturing, 28(1), 87–97, 2015. http://doi.org/10.1080/0951192X.2014.941937.

[23] Mullins, S.; Anderson, D.: Automatic identification of geometric constraints in mechanical assemblies. Computer-Aided Design, 30(9), 715–726, 1998. http://doi.org/10.1016/S0010-4485(98)00026-8.

[24] Mun, D.; Han, S.: Identification of topological entities and naming mapping for parametric CAD model exchanges. International Journal of CAD/CAM, 5(1), 69–82, 2005.

[25] Piegl, L.A.: Ten challenges in computer-aided design. Computer-Aided Design, 37(4), 461–470, 2005. http://doi.org/10.1016/j.cad.2004.08.012.

[26] Shah, J.J.; Mäntylä, M.: Parametric and feature-based CAD/CAM: concepts, techniques, and applications. John Wiley & Sons, 1995.

[27] Staves, D.R.; Salmon, J.L.; Red, W.E.: Associative CAD references in the neutral parametric canonical form. Computer-Aided Design and Applications, 14(4), 408–421, 2017. http://doi.org/10.1080/16864360.2016.1257184.

[28] Vilmart, H.; Léon, J.C.; Ulliana, F.: From CAD assemblies toward knowledge-based assemblies using an intrinsic knowledge-based assembly model. Computer-Aided Design and Applications, 15(3), 300–317, 2018. http://doi.org/10.1080/16864360.2017.1397882.

[29] Wang, Y.; Nnaji, B.O.: Geometry-based semantic ID for persistent and interoperable reference in feature-based parametric modeling. Computer-Aided Design, 37(10), 1081–1093, 2005. http://doi.org/10.1016/j.cad.2004.11.009.

[30] Wesley, M.A.; Lozano-Perez, T.; Lieberman, L.I.; Lavin, M.A.; Grossman, D.D.: A geometric modeling system for automated mechanical assembly. IBM Journal of Research and Development, 24(1), 64–74, 1980. http://doi.org/10.1147/rd.241.0064.

[31] Wilson, R.H.; Latombe, J.C.: Geometric reasoning about mechanical assembly. Artificial Intelligence, 71(2), 371–396, 1994. http://doi.org/10.1016/0004-3702(94)90048-5.

[32] Zhang, W.; Ma, M.; Li, H.; Yu, J.: Generating interference matrices for automatic assembly sequence planning. The International Journal of Advanced Manufacturing Technology, 90(1-4), 1187–1201, 2017. http://doi.org/10.1007/s00170-016-9410-x.

[33] Zhao, Y.F.; Horst, J.A.; Kramer, T.R.; Rippey, W.; Brown, R.J.: Quality information framework–integrating metrology processes. IFAC Proceedings Volumes, 45(6), 1301–1308, 2012. http://doi.org/10.3182/20120523-3-RO-2023.00113.