CAD-CAE Integration for Composite Laminate Design Optimization

Zijun Wu, Wei Liu, Xiaojie Wu, Qinghua Liu, and Zhengdong Huang
Huazhong University of Science and Technology, Wuhan, P.R. China
Email: wuzijun2010@gmail.com, justyyyo@126.com, stepsforward@163.com, {liuqh, zdhuang}@hust.edu.cn

***Abstract*—CAD-CAE integration is an important issue for complex structure optimization, which automatically transforms the geometric model from CAD systems to CAE systems and maintains the associations of different types of information after design variables are updated. In composite laminate design optimization (CLDO), the design variables include not only geometric dimensions but also fiber orientation angles and laminate thickness of different plies at individual middle surfaces of a compound shell structure, which makes the information association maintenance more difficult. This paper proposes a CAD-CAE integration scheme for CLDO. In this scheme, an analysis task description for parametric CAD model is created to guide the automatic FEA modeling and analysis process in CAE systems on the basis of parameter-independent identifiers for geometric entities. The permanent naming for geometric entities shared by both CAD and CAE systems is achieved with a marking-point approach and it helps maintaining the association relations between geometric objects and their physical information for different parameters. The proposed approach has been successively applied to create the automatic procedures for static and modal analyses of a complex parameterized shell structure in its CLDO.**

***Index Terms*—CAD-CAE integration; composite laminate; material direction; permanent naming**

I. INTRODUCTION

The design of mechanical products involves two kinds of software, CAD systems for product definition and CAE systems for performance analysis. However, it is not easy to make them work together because they have relatively independent geometric models. Currently, two approaches are mostly used for integrating the two types of software: CAD-centric and CAE-centric. The former integrates some CAE modules in the CAD platforms while the latter develops some CAD functions in the CAE systems. But the two approaches restrict the user's preferences for certain CAD or CAE systems. Especially for the purpose of multi-disciplinary design optimization, which requires various CAE systems, the integration based on data transfer is more flexible for practical applications [1]. Nevertheless, it is hard to create the data interface for this integration, especially when the data vary with the design variables in design optimization.

There are many scholars who have conducted the research on the problem of CAE-CAE integration. Nosenzo *et al*. [2] presented a case study on CAD-CAE integration in a PLM environment and raised some questions regarding management issues on design models and simulation scenarios as well as their relationships. Graignic *et al*. [3] developed a software framework for managing the various analysis models in complex multidisciplinary system design, in which the model data contexts are elaborated to reflect the interactions between different behavior simulations, including those between different components, system levels or physical domains. Byoung-Keon *et al*. [4] proposed a sharable format for multidisciplinary FEA data in a collaborative design process, which can significantly reduce the sizes of files to be transferred and facilitate improving the efficiency of the integration, including that between CAD and CAE.

Besides the data transfer and management issue, geometric conformality is another important concern for CAD-CAE integration. Lee [5] extended the traditional feature-based modeling technique to represent the geometric models for CAE, which have geometric elements with various dimensions composed in non-manifold topology. Sypkens Smit and Bronsvoort [6] proposed a geometric modeling approach for CAE by adding analysis views to multiple-view feature models in CAD systems. Furthermore, Hamri [7] presented a mixed shape representation for CAE, which not only supports the precise B-Rep in manifold and non-manifold topologies but also covers the approximate polyhedral models. In the method, the different type models are maintained on the same topology called the High Level Topology (HLT). While the representation approaches mentioned above are all for a single body or a part, Zeng *et al*. [8] have addressed the geometric representation issue for the analysis of multi-body or assembly, where an Analysis Building Block model (ABB) is utilized to capture analytical engineering information related to different geometric entities. To produce the above geometric representations for CAE from CAD models, some adaptations or simplifications of CAD models are required. Foucault *et al*. [9] proposed a topology adaptation approach for CAD models to meet the requirement in generating qualified finite element meshes.

Manuscript received June 1, 2015; revised September 11, 2015.

doi: 10.18178/ijmerr.4.4.373-383

Thakur *et al*. [10] have presented a review on geometric simplification methods for CAE analysis from CAD models. Since the simplified geometric models for CAE are usually composed of entities with mixed dimensions, the analytical coupling between the meshes with different dimensions has also been studied by some scholars [11]. Although the geometric adaptation could be conducted in CAD systems, current industrial practice prefers to accept it as a step of the pre-process in CAE systems.

Design optimization poses more strict requirements on CAD-CAE integration because the full automation is expected from design to analysis, including the automatic FEA model creation driven by geometric change. In order to realize the design iterations based on repeated simulations, Haimes and Merchant [12] have studied the enabling features of software foundation for tightly integrating the CAD system to the downstream analysis tool. Van der Velden [13] developed a component-based architecture in iSIGHT-FD to automatically propagate changes in CAD to CAE systems. Gujarathi and Ma [14] proposed a common data model to accommodate the required parametric information for both CAD modeling and CAE analysis, which facilitates maintaining the associative dependences among them. An obvious shortcoming of the mentioned research on the integration for design optimization is that they didn't elaborate on the detailed approach for automatic FEA model generation driven by design changes. Actually, design changes heavily affect the operation commands as well as process flows for analysis model creations, especially when the topology change and location change for constraints and loads are present. This paper studies the CAD-CAE integration for composite laminate design optimization (CLDO), in which the analysis model creation is a more complex procedure because mid-surface extraction and fiber directions on different plies should be considered.

Composite laminated structures are widely used in aerospace industry due to their high stiffness and strength to weight ratios. Since there is a possibility of tailoring their stiffness and strength by selecting fiber orientations, composite laminate design optimization has received a lot of attention in the last decades. Many researchers addressed the combinatorial problem in the optimization and some discrete optimization algorithms like Genetic Algorithms were proposed [15]. In order to use the algorithms with higher efficiency, some scholars converted the discrete problem into a continuous one by relaxing its design variables and choosing an appropriate parameterization for the extended design space [16], [17]. Nevertheless, its computation efficiency still remains a problem and hence some metamodeling techniques are adopted to reduce the times of analyses [18], [19]. As the efficiency issue is able to be handled with various approaches, the practical application of CLDO is expected. However, except for a recent work on the information management for laminated composites with a semantic approach [20], which benefits the data transfer across different software systems, the current research on CLDO rarely considers the CAD-CAE integration problem though the integration is important for designing a composite laminated structure with realistic complexity.

Actually, the design variables in CLDO include not only geometric dimensions but also fiber orientation angles and laminate thickness of different plies at individual middle surfaces of a compound shell structure, which makes the information association maintenance more difficult. This paper proposes a CAD-CAE integration scheme for CLDO. In this scheme, an analysis task description for parametric CAD model is created to guide the automatic FEA modeling and analysis process in CAE systems on the basis of parameter-independent identifiers for geometric entities. The permanent naming for geometric entities shared by both CAD and CAE systems is achieved with a marking-point approach and it helps maintaining the association relations between geometric objects and their physical information for different parameters. The proposed approach has been successively applied to create the automatic procedures for static and modal analyses of a complex parameterized shell structure in its CLDO.

The paper is organized as follows. Section 2 gives the overview of the proposed CAD-CAE integration scheme and Section 3 introduces a marking-point approach for naming the geometric entities shared by both CAD and CAE systems. Based on the parameter-independent identifiers for the geometric entities, an analysis task description method is developed in Section 4. After this, Section 5 discusses about building parametric procedures for FEA modeling and analysis according to the analysis task description. To validate the proposed approach, a composite laminate analysis example is presented in Section 6. Finally, the paper is finished with some conclusive remarks in Section 7.

II. Overview of the CAD-CAE Integration Scheme

The main task of CAD-CAE integration is to prepare appropriate input data for CAE in CAD systems and to conduct the analysis task automatically driven by the input data in CAE systems. To this end, we propose a CAD-CAE integration scheme in this paper as shown in Fig. 1, which includes the following components:

- Extend the parametric CAD model to accommodate analysis information. Firstly, starting off the feature-based CAD models, some geometric entities that represent locations for constraining, loading, connecting, locating/orientating and checking, called analysis features, are added. Secondly, a set of points are inserted to the model as well to mark the analysis features and the geometric entities for analysis domain definition, which are also called analysis features with a type of beam, shell, or solid though they may are all created as a solid in the CAD model. Finally, an analysis task description (ATD) file is created to present the analysis type, domain definition, load case and output required with the analysis features.

- Develop the automatic procedures that conduct FEA modeling and analysis for the parametric CAD model. Based on the CAE support environments [20] like HyperMesh, Ansys, Nastran and Abaqus, etc., some automatic procedures are created with various application development tools; the procedures includes geometry import, analysis feature identification, feature geometry extraction, meshing, loading, solving, and result extraction. Since the procedures are completely driven by the parametric CAD model and its ATD file, their complexity is closely related to the representative ranges of the two files. Here, the static, modal and buckling analyses for composite laminates are focused.
- Create the interfaces in the optimization software, which drive the CAD model update and CAE analysis in sequence. After the parametric CAD model is extended and the CAE procedures are developed, they are controlled by the optimization software by means of passing the design parameters ***x*** and extracting the design performance evaluations from the Analysis Result Report (ARR) file. For this purpose, the software is supposed to provide the interface functions to handle the I/O files.

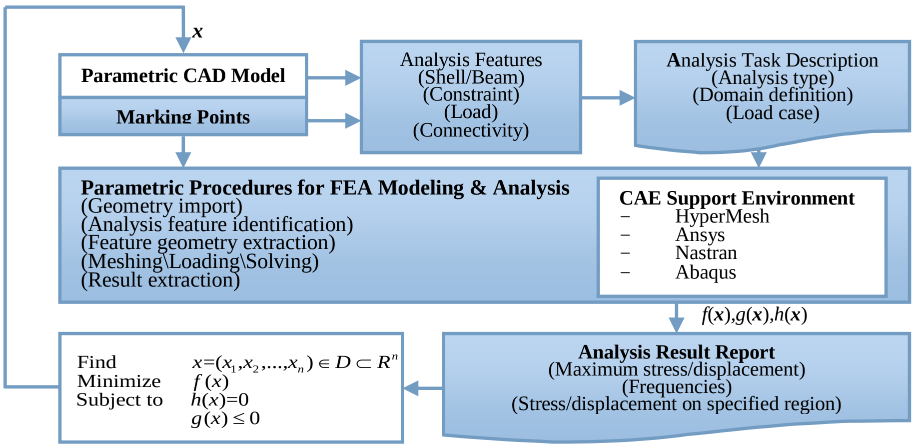


Figure 1. The proposed framework for CAD-CAE integration.

III. Parameter-Independent Identifiers for Analysis Features

Traditionally, CAD systems build models that only contain the geometric information and not consider the information for analysis. In the proposed approach, we extend the models to include analysis information, which is organized in the models as analysis features. However, the feature information is usually lost during the operation of geometric model import in CAE environments; the import in most CAE systems can only obtain non-parametric B-rep information. Thus, the analysis information created in CAD systems cannot be shared by CAE systems. It is worthwhile to note that creating analysis information in CAE environments usually results in repeated FEA modeling for different design parameters.

To overcome this problem, this paper develops an approach for identifying the analysis features in CAE environments. In the approach, the original design-feature based geometric model is first augmented by adding analysis features. Then, to make the analysis features can be identified without using the texts attached to the geometric entities, which may not be kept after format transformation or data transfer, some marking points are added to the analysis features (see Fig. 2). The marking points are a set of 3D points that are located on the geometric entities for an analysis feature; for example, they are chosen as the point at a vertex, the midpoint of an edge or an interior point of a face. In addition, the coordinates of the marking points should be parameterized like a feature in order to make them change with their associated geometric entities after the parameters are updated. Therefore, the marking points for individual analysis features can also be organized as a feature in CAD systems.

As shown in Fig.2, when the parameter x is changed, the extended parametric CAD model can be updated by rebuilding its B-rep model with the API or GUI command of CAD systems. After this, the new B-rep model and the new coordinates of marking points with labels or IDs are obtained. Since the analysis task description is created by means of analysis features, where the analysis features are quoted via their identifiers same as the labels for their associated marking points, the analysis feature identification program developed in CAE systems can find out the geometric entities for an analysis feature through matching its associated marking points with vertices, edges, faces and solids in the new B-rep model. Obviously, the identifiers for analysis features are independent of model parameters. Here, the extended parametric CAD model and its parametric analysis task description, which are generated in advance, both keep unchanged for different value settings to the model parameters, but the new B-rep model, the marking point coordinates and the ATD file vary with changes of the parameter values; thus, the analysis feature geometry obtained in CAE may be different for different parameter values.

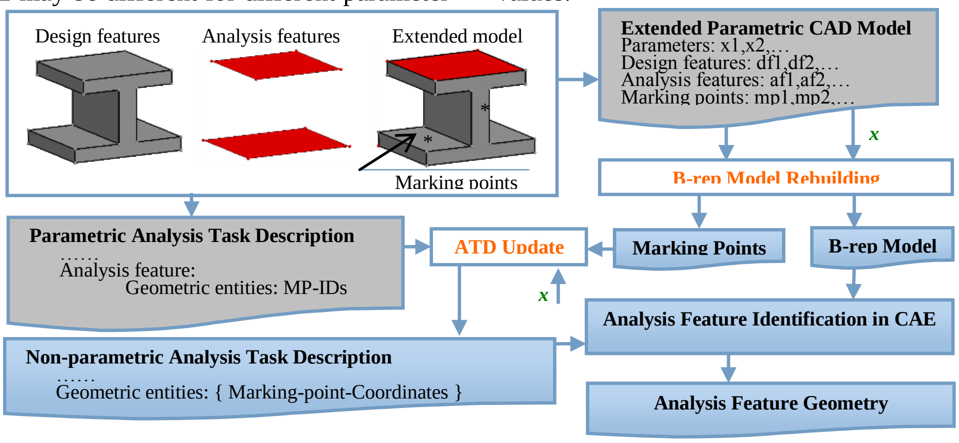


Figure 2. Analysis feature identification with marking points.

IV. Analysis Task Description Based on Analysis Features

The extended geometric model contains the geometric information that is needed for FEA modeling, but it is not enough for the analysis model construction. For example, the information of the loaded force direction and size is missed and the material distributions on different domains are not explicitly given in the geometric model. Another problem is that the interrelationship between the added domains cannot be delivered to CAE systems along with the CAD-CAE data transfer operation.

To address the above problems, we define a text file, which is called Analysis Task Description file (ATD, see the example in Appendix A), to transfer the missed information to CAE systems. The ATD file has two forms, parametric and non-parametric, as shown in Fig. 2. The parametric ATD file has some variables in the expressions for its analysis parameters like material directions and properties, and it can be created together with the extended parametric CAD model before the optimization is carried out. The non-parametric ATD file has a fixed value for all its analysis parameters, and it also works with non-parametric B-rep model to form a complete primary data of the FEA modeling for a fixed design. In spite of the difference, the two types of ATD files have the same information structure, which is presented in Fig. 2.

Mainly, the ATD file includes information about domains, connects, constraints, loads, checks, material properties and tasks. A domain usually corresponds to a domain feature like a lumped mass point, a shell or a solid represented with a CAD part. If the part model for a lumped mass point or a shell is a solid, it will be simplified to a point or a mid-surface in CAE systems according to its feature type definition. A shell with composite materials has a complex material distribution. To handle this issue, material distribution feature is introduced and a set of this type features are attached to a domain feature in order to describe the detailed material definitions for the laminates on the various regions of the shell. A connect represents a rigid or flexible connection between two domain features. To specify its connection location, connecting features are introduced here. Similarly, a constraint, load and check respectively have constraining features, loading features and checking features to describe their acting locations. Material properties are presented with three levels of entities Property, Laminate and Material; they respectively focus on the physical and material parameters of volume feature level, material distribution level and point-wise level. Specially, a laminate entity includes a material system feature to express the geometric information for the material coordinate frame. In addition, the material properties include some material parameters that could serve as design variables in the optimization. The ATD file is formed on the basis of analysis features; Fig. 3 has listed all the analysis feature types. All the analysis features appearing in the file should have their geometric definitions in the extended parametric CAD model and the marking points for their naming across CAD and CAE systems should have also been defined. In this research, the marking points are stored in the ATD file as well. Following are the some other characteristics of the ATD file:

- For the convenience of identification, each entity has its own string-type identifier though its association with geometry entities needs to be created via the marking points in CAE systems.
- In order to avoid ambiguity, the marking points for an entity (e.g. a face) must not be the points on its boundary (e.g. an edge of the face). They must be an interior point of the entity.
- The material plies in a laminate entity must be defined independently and the order in which they are listed is just their stacking sequence in the composite material. The detailed definition will be described in Section 5.3.

It should be noted that the ATD file only provide the analysis information for creating a FEA model and it doesn't give the detailed geometric information, which is still provided in the CAD file. The ATD file is a bridge between CAD and CAE systems in our proposed integration method, which is output from CAD systems and input to CAE systems as shown in Fig. 2.

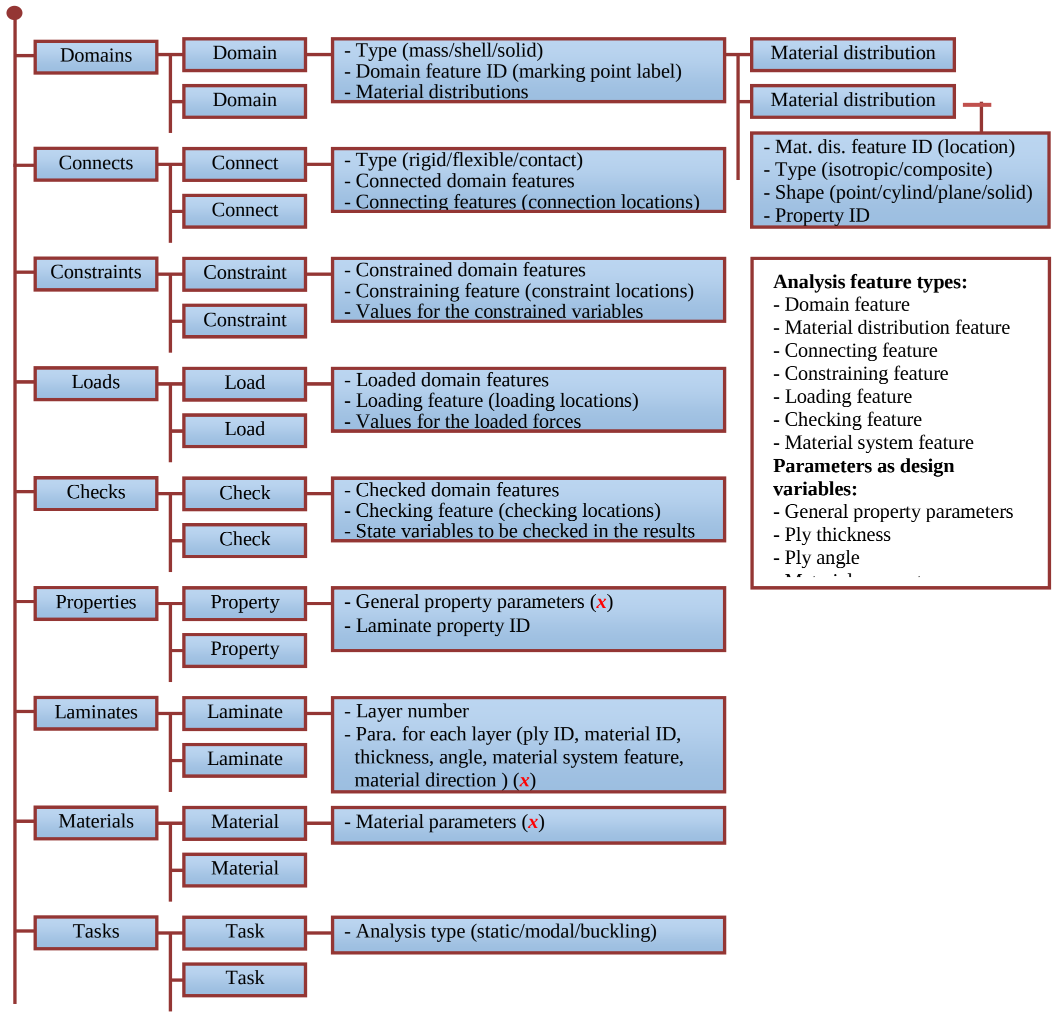


Figure 3. The structure of the analysis task description file.

V. Procedures for FEA Modeling and Analysis

Traditional CAE systems pay more attention to the man-machine interaction based on the visualization technology. However, the automatic processes for geometric model update, FEA modeling and analysis become more and more important in order to meet the requirement of design optimization. Here, the geometric model update includes the B-rep model rebuilding and ATD update in CAD systems (see Fig. 2), which is implemented with the APIs of the specific CAD systems. In this section, we describe the method for realizing the automation in building FEA models and performing the analysis task in CAE systems on the basis of the analysis task description file. This is also called the encapsulation of CAE modeling and analysis in this paper. In our proposed integration approach, the automation is performed on geometry import, analysis feature identification, feature geometry extraction, meshing, loading, and solving and result extraction. Here, we only focus on some of them that need special treatments.

A. Geometry Information Identification

Geometric information identification includes two aspects, analysis feature identification and feature geometry extraction. In the analysis feature identification, a set of geometric entities associated with a specific analysis feature is extracted from the geometric model imported into CAE systems. Since each analysis feature has a set of marking points, which can be obtained from ATD file, its geometric entities can be identified by searching for the entities with the minimum distance to the points. The entities could be solids, surfaces, curves or points, depending on the feature types.

Although, the geometric entities for a feature can be obtained in analysis feature identification, a further process is still needed to extract the geometric information that has special meaning for the feature. For example, the middle surfaces should be extracted from its solid geometric entities for a domain feature with a type of shell. This process is called feature geometry extraction here. Actually, the middle surface extraction plays an important role in this research regarding the analysis of composite laminate structures and the quality of the generated middle surfaces has important impact on the quality of meshes and the accuracy of the numerical result. Generally, it is not easy to develop a robust procedure for automatic middle surface extraction from a complex solid structure. But some CAE systems like HyperMesh can provide such functions with pretty good results for solid models that have appropriate structures and dimensional proportions.

B. Automatic Meshing with Quality Control

Here, we only discuss the automatic meshing for surfaces because all the domains in the analysis of composite laminate structures are the middle surfaces. To achieve a mesh quality required by the FEA solvers to produce an acceptable numerical result, the mesh quality control is needed. Usually, a mesh quality measure, called Quality Index (QI), is used to evaluate the generated meshes. When QI is less than a given positive value A (it is set to 0.03 in this research), the mesh is thought to be acceptable. For a complex structure, it is almost impossible to achieve this goal with only one meshing operation. Therefore, some re-meshing operations are required. It is a common practice in industry to perform the re-meshing operations interactively under the help of mesh quality visualization. However, this is not applicable to the analysis for optimization. To handle this problem, an iteration procedure for re-meshing is developed. In the procedure, the regions in the current mesh that fail to meet the quality requirement are extracted first, and then the meshes for these regions are re-generated; the process is iterated until all the meshes satisfy the QI condition. Fig. 4 presents the flowchart of the procedure and a mesh result for an I-beam structure.

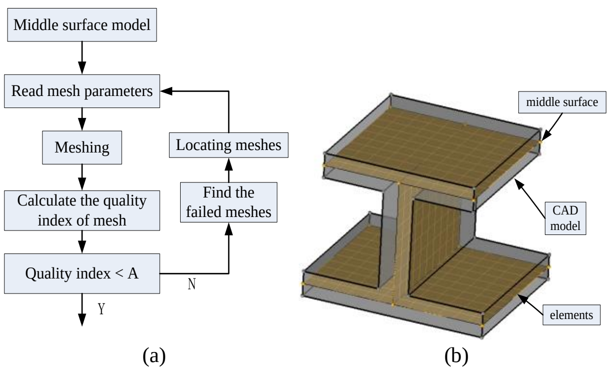


Figure 4. Re-meshing procedure with quality control. (a) Iterated meshing process. (b) The mesh generated for I-beam.

In the mesh generation phase, the meshing operations are only performed to domain features, including shells and lumped mass points here. For the lumped mass points, some special elements are created in CAE systems. For other analysis features like loading features, no meshes are generated because their geometric entities only serve as representations of some specific locations. In addition, to provide the means for the users to control the meshing operation, some mesh parameters such as mesh types and mesh sizes can be set in the ATD file.

C. Constraint and Load Definition in Analysis Model

Since constraining features, loading features, connecting features and checking features have already been defined in CAD systems and imported into CAE systems via the ATD file, the FEA modeling here only needs to assign the feature information to the meshes. For this purpose, the geometric entities representing the locations of the features are used to search for the corresponding elements in the meshes. After the elements are found out, the feature parameters like the values for the constrained state variables and loaded forces can be set to the elements. For connecting features, some rigid or flexible connection elements are created between the nodes of the identified elements. For checking features, the corresponding state variables at the nodes of the identified elements are recorded for the retrieval of the computation results after the analysis is finished.

However, it is not an easy task to find the elements out of the meshes with thousands of elements. First, since the analysis features are defined with faces of the original solid models, the geometric entities for a feature are not directly located on the meshes of the middle-surfaces. So, the element search is to find the elements that are parallel to the solid faces for the features and have the nearest distance at the same time. Second, finding the nearest parallel elements is a time-consuming process. To identify the mesh with a given distance to the feature faces, we follow the steps: (1) calculate the distances of three random nodes of an element to all the feature faces; (2) find a feature that has three equal or approximately equal distances to the element; (3) if the distance is minimum among all the features, then this feature is the nearest parallel one for the current element. It is worth noting that the distances may not be a vertical distance for the faces. If the node projection on a face is within the boundary of the face, the calculated distance is vertical. If not, the distance is the length of the line connecting the node and the nearest point on an edge of the face as shown in Fig. 5. In the figure, E is an element, and d_{11},d_{12}, d_{13} and d_{21},d_{22}, d_{23} are the distances from three nodes of E to two planar faces P_1 and P_2, respectively.

Classification on the elements is a critical step to improve the efficiency of the analysis model creation in CAE systems. In the above step, each feature has its own corresponding nearest elements. In this step, we read the identifier of each feature and save the indices of its corresponding nearest elements in a feature-elements table. The table greatly facilitates the information assignment to or state extraction from the corresponding elements of analysis features.

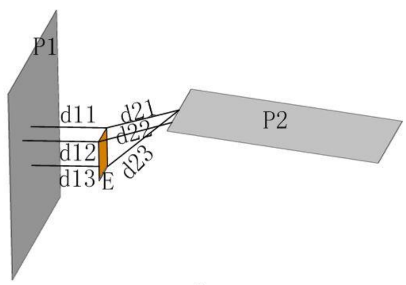


Figure 5. The distance between an element and a feature face.

D. Defining Composite Materials

The material definition for a composite material structure is much more complicated than those for ordinary isotropic material objects because it involves handling geometric information. There are three tasks for the composite material definition: (1) define the configuration of composite laminate panels on the analysis domain; (2) define the stacking sequences of composite laminates for each panel; (3) define the material fiber direction for each layer of the laminates. The first task is to create a region partition to the shells represented by the middle surfaces. Actually, the region partition information is specified by the structure designers in CAD systems and is saved in the ATD file as a set of material distribution features (see Fig. 3). For each material distribution feature, which represents a composite laminate panel, there are a set of marking points that help us to identify the feature faces and the corresponding mesh elements as well on the middle surfaces in the way described in Section 5.3. In CAE systems, every panel has a material property entity defined and the property ID is assigned to each elements in the panel. Furthermore, the second task can be carried out by directly assigning the ply order specifications in the ATD file to the properties of the defined panels. But, due to the fiber directions vary for different laminates, it is hard to manage with a program though they can be adjusted easily in visualization systems.

In the proposed CAD-CAE integration method, we assume that the directions of fibers for each ply are the same; this means that we do not consider the curvilinear fibers [19] in this research. As shown in Fig. 6, the fiber directions are first defined on the geometric model by the structure designers and then they are transformed to an expression related to individual mesh elements. Usually, there are several material coordinate systems defined in the geometric model, and a material reference direction is chosen for each composite laminate panel from one of the systems as its x-axis, y-axis, z-axis, or r-axis when it is looked as a cylindrical coordinate system. Then, the fiber direction is defined via the angle θ from the material reference direction to the fibers. However, for the FEA analysis, the angle information should be transferred to the mesh elements of the panel. In an element, the fiber direction *f* is defined through two vectors: the pre-defined material reference direction *x* and the normal *n* of the current element, which are shown in Fig. 6(b). The vector y in the figure is got through the right-hand rule from n to *x*, and the fiber angle is just $\theta = \pi/2 - \alpha$, where α is the angle from y to the fiber direction f. But, the normal of an element might be randomly chosen by CAE systems as the opposite direction because a plane element has two opposite normal vectors. If the normal is the other one, the above method will define a different fiber direction f. To overcome this dilemma, we utilize the locations of the marking points for the material distribution features in the ATD file, which is introduced in Section 4.

The method for determining the element normal (or middle surface normal) is shown in Fig. 7. The middle surface is displayed in yellow color while the elements for the middle surface are in nattier blue. The black point is the marking point defined in the ATD file and the red point with blue circle is its projection on the middle surface. Here, the normal of the middle surface is chosen as a vector pointing to the side in which the marking point is located. This means that the normal has an acute angle with the vector from the projection point and the marking point. If the marking point is defined on a face in the other side of the middle surface, the normal is reversed.

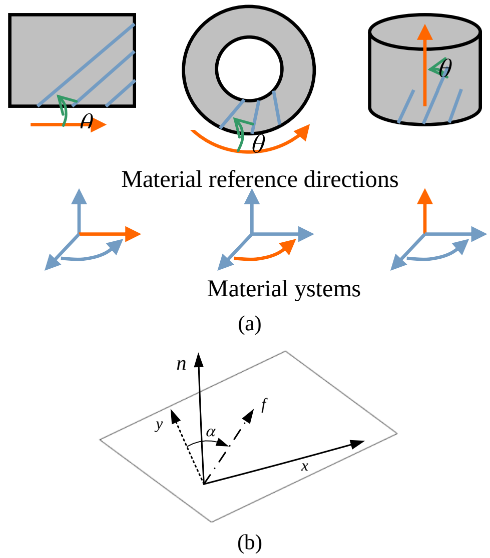


Figure 6. The definition of fiber directions. (a) The fiber direction definition on the geometric model; (b)The fiber direction definition on the mesh elements

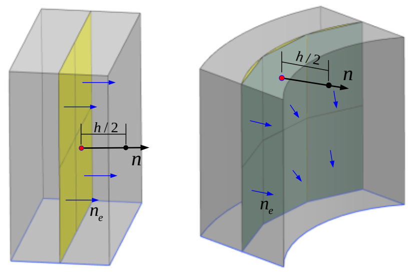


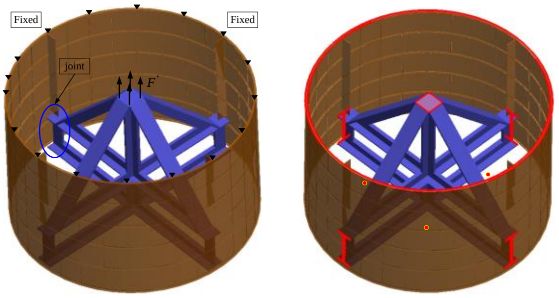
Figure 7. Element normal determination from the locations of marking points.

VI. EXAMPLE

To testify our proposed CAE-CAE integration method, we apply it to the analysis of a complex shell structure with composite material. Through the FEA modeling and analysis for this structure, the accuracy of middle surface extraction and the reliability of the automatic meshing are verified. In addition, the methods for constraint, load and connection definitions and the approach for fiber direction calculation are also examined. In this example, we use Pro/E to build the geometric model and HyperMesh to construct the analysis model. To demonstrate the integration with more software systems, Nastran and Abaqus are selected as solvers respectively for its modal and static analyses.

The structure is a cylindrical cabin with strengthening bracket, which is composed of two components: the rood beam and the thin cylindrical wall shown in Fig. 8(a). For the analysis, the bottom of the cylinder is fixed and the small top planar face of the beam is loaded with a vertical downward distributed force. In addition, these two components are joined through the four stretched legs of the cross-shaped beam.

Firstly, some accessory geometric entities like those representing the boundary condition regions are created in the CAD model. In this example, we also define extended geometric entities for the rigid joints and the distributed force, which are shown in Fig. 8(b). Besides these, some lumped mass points are created as well to represent the equipments supported by the bracket, which are shown as the red points with yellow circles in the same figure.



(a) (b)

Figure 8. Cylindrical cabin with strengthening bracket. (a) The model with boundary conditions. (b) The extended geometric model.

Secondly, the analysis task description files respectively for the modal analysis and static analysis of the structure are generated. In this step, many material distribution features are also defined to express different composite laminate panels on both the beam and the wall components, and each of the features has its marking points being created for its identification in CAE systems. Accompanying with these material distribution features, ply stacking sequences and fiber directions are also specified.

Thirdly, the middle surfaces are extracted from the CAD models in Fig. 8 and then their finite element mesh is generated with user application programs developed in HyperMesh, implementing the procedures discussed in Section 5. The mesh is shown in Fig. 9, which has 83266 elements.

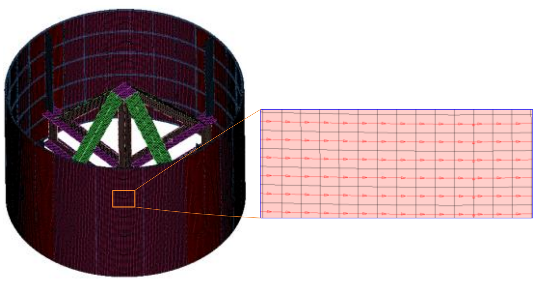


Figure 9. The finite element mesh and its fiber paths.

Fourthly, the constraints for the FEA model are imposed with the programs via the definitions in the ATD file, including the boundary conditions, the external force and the rigid connects between the wall and the bracket. Finally, the property entities for composite material in laminate panels are created and attached to the FEA model in the CAE system, during which the material fiber directions can be automatically confirmed with the approach discussed in Section 5. The FEA model and its partial fiber directions are presented in Fig. 9.

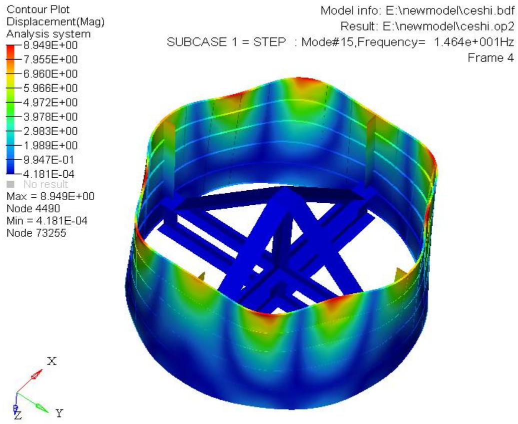
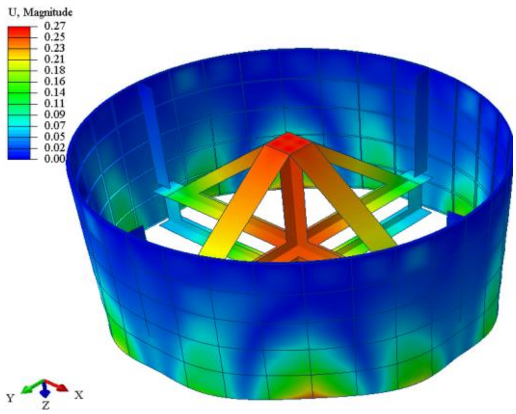
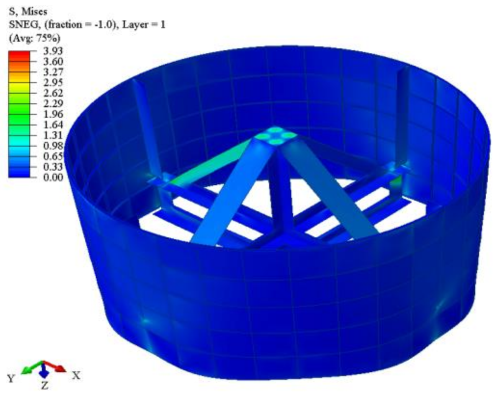


Figure 10. The 15th mode shape for the example.

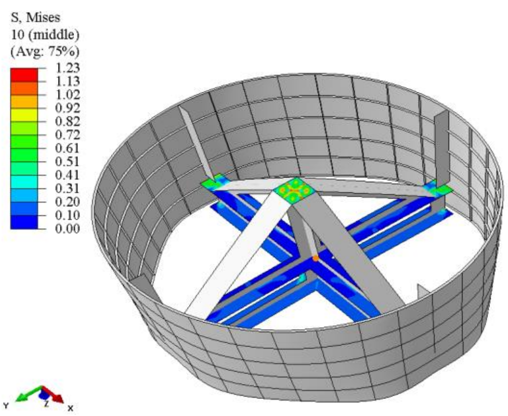


(a)



(b)

Figure 11. The static analysis result for the example. (a) The displacement contour. (b) The stress contour



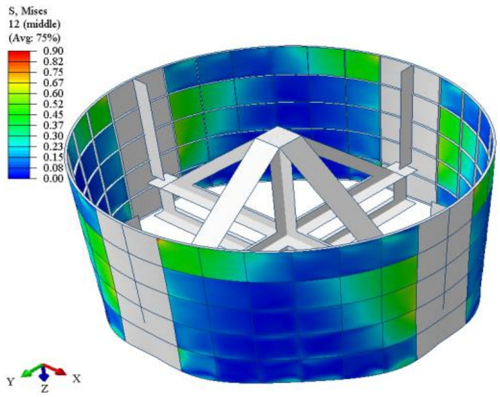


Figure 12. The stress on the 10th and 12th plies in the static analysis.

After constructing the analysis model, the Nastran and Abaqus solvers are started to respectively carry out the modal analysis and the static analysis. The results of the analyses can be visualized with HyperView. Fig. 10 presents the 15th mode shape obtained in the modal analysis. Fig. 11 shows the contours of global displacement and stress from the static analysis. In this example, there are 13 plies of composite laminate material. The static stress contour for the10th and 12th plies are shown in Fig. 12. On the cylindrical wall, due to the ribbed slabs, the distribution of stress appears grid-shaped.

VII. Conclusions

In this article, we have proposed an approach for CAD-CAE integration based on geometric association with marking-points to support the composite laminate design optimization. In the approach, the CAD model is extended to include analysis features, which are utilized to express the boundary conditions, loading forces, connection constraints, checked state variables and local material distributions in the same parameterized framework as design features. To convey the information to CAE systems, an analysis task description based on analysis features is developed to express the overall associations between entities related to the analysis task. In the proposed integration method, the ATD file is important because it is the bridge between CAD system and CAE system and provides a channel for sharing the information between geometric model and analysis model. However, the ATD file is only a higher-level description for a FEA model; although it facilitates the FEA model update for parameter change, the correspondences between the geometric entities for different parameters should be carefully maintained. To address this issue, a marking point approach is developed here to identify the corresponding entities. Particularly for the composite laminate analysis, some methods for the definitions of laminate panel, material system and fiber direction are elaborated in this paper. Taking the ATD file and the extended CAD model as input, several automatic procedures are developed as user application programs in a CAE environment to perform the FEA modeling and analysis. The proposed CAD-CAE integration method for CLDO performs well for the complex test example, which shows that it is very promising for such kind of applications.

Mainly, the proposed integration method has three advantages. Firstly, the method supports the multi-disciplinary optimization which requires the computations on various CAE systems; this is more flexible than the CAD-centric and CAE-centric integrations. Secondly, the parameterization for analysis information is accomplished with the same software tools for CAD models, which is convenient for designers. Thirdly, it can effectively handle the association maintenance issue for the geometric entities between different software systems; this makes it possible to optimize the composite laminate structures that have plenty of physical information related to specific geometry. Nevertheless, there still exist some topics that need to be investigated in the future, such as the robustness improvement of the automatic procedures for geometry identification and simplification, the extension of analysis types to cover a wider range of CAE tasks, the usage of engineering knowledge in FEA modeling, and the extension of material types to include composite material with curvilinear fibers. Here, the proposed integration method is only tested with the integration among the systems of Pro/E, HyperMesh, Nastran and Abaqus through the analysis of the cylindrical cabin with strengthening bracket, and we have not examined its effectiveness on other software and the problem with isotropic material. Despite this, we still believe that the present work provides one of potential directions for solving the problem of CAD-CAE integration.

Appendix A

The following is an example of ATD file:

```
# This is a CAE task description file for CAD model
task-begin
    task-type       LINEAR-STATIC
task-end
# Domain features
parts-begin
    part HUST-1     SHELL      hcsProp     NULL NULL
    part HUST-2     SHELL      hjProp      NULL NULL
parts-end
# Material distribution features
features-begin
    feature         HUST-1    cyFaces     CYLIND   cyFacesPNT
                    cyFacesProp           matCsys1  3-axis
    feature         HUST-1    cyFacesU    CYLIND   cyFaceUPNT
                    cyFacesProp           matCsys1  3-axis
    feature         HUST-2    HPlanes     PLANE    HPlanesPNT
                    HPlanesProp           matCsys1  1-axis
    feature         HUST-2    VPlanes     PLANE    VPlanesPNT
                    VPlanesProp           matCsys1  1-axis
features-end
# Marking points
faceSets-begin
    face-set        cyFacesPNT  (834.1065,-167.2030,-726.5380)
                    CIRCLE (0.0,0.0,0.0)    48   7.5
    face-set        cyFaceUPNT (-618.6969,578.0268,-869.7862)
                    CIRCLE (0.0,0.0,0.0)     2    180.0
    ……
faceSets-end
# Connect features
rigid-connects-begin
    rigid-connect   connect1   HUST-2    HUST-1    Loc1  Loc2
```

```
    rigid-connect    connect2   HUST-1    HUST-2    Loc3  Loc4
rigid-connects-end
# Materials
materials-begin
    material  baseMat       MAT1  (2.0e5,0.0,0.35,1.8e6,0,0,0,0,0,0,0)
    material  carbonFiber  MAT8  (290.0,290.0,0.30,1300.0,0.0,0.0,
                                            0.0,0.0,0.0,0.0,0.0,5650.0,5650.0,
                                            5650.0,5650.0,1000.0,0.0,0.0,0.0,0.0)
materials-end
# Material systems
material-csys-begin
     matCsys          matCsys1  (0,0,0;1,0,0;0,1,0;0,0,1)
materials-csy-end
# Laminates
laminate-materials-begin
    laminate          cyFacesProp   3
          laminate-layer              1   1          carbonFiber  0.2  45.0
          laminate-layer              2   2          baseMat       1.2  90.0
          laminate-layer              3   3          carbonFiber  0.2  -45.0
    ……
laminate-materials-end
# Constraint features
displacement-constraints-begin
    displacement-constraint     HUST-2 constr  NULL  0 0 0 0 0 0
displacement-constraints-end
# Loading features
plane-pressures-begin
    plane-pressure                  HUST-1  force  NULL  -1   0.2
plane-pressures-end
#
```

ACKNOWLEDGEMENTS

This project is supported by National Natural Science Foundation of China (Grant Nos. #61173115, #51475186 and #51375186). The authors would like to thank Mingxi Zhang and Jiani Zeng for their contributions on the computer programming and case study in this project.

REFERENCES

[1] S.Gordon, “An analyst’s view: STEP-enabled CAD–CAE integration,” in *Proc. Presentation Materials of NASA’s STEP for Aerospace Workshop*, Pasadena, CA 2001, pp. 16–19.

[2] V. Nosenzo, S. Tornincasa, E. Bonisoli, and M. Brino, “Open questions on Product Lifecycle Management (PLM) with CAD/CAE integration,” *International Journal on Interactive Design and Manufacturing*, vol. 8, no. 5, pp. 91-107, May 2014.

[3] P. Graignic, T. Vosgien, M. Jankovic, V. Tuloup, J. Berquet, and N. Troussier, “Complex system simulation: proposition of a MBSE framework for design-analysis integration,” *Procedia Computer Science*, vol. 16, pp. 59-68, 2013.

[4] P. Byoung-Keon and J. J. Kim, “A sharable format for multidisciplinary finite element analysis data,” *Computer-Aided Design*, vol. 44, no. 7, pp. 626-636, July, 2012.

[5] S. H. Lee, “A CAD-CAE integration approach using feature-based multi-resolution and multi-abstraction modeling techniques,” *Computer-Aided Design*, vol. 37, no. 9, pp. 941-955, August 2005.

[6] M. S. Smit and W. F. Bronsvoort, “Integration of design and analysis models,” *Computer-Aided Design and Applications*, vol. 6, no. 6, pp. 795-808, 2009.

[7] O. Hamri, “CAD-CAE integration through a mixed shape representation,” in *Proc. ASME 2011 IDETC/CIE*, Washington, 2011, pp. 28-31.

[8] S. Zeng, A. Xiao, and R. S. Peak, “GeoTran-HC: Geometric transformation of highly coupled variable topology multi-body problems,” *Computer-Aided Design*, vol. 39, no. 9, pp. 756-771, September 2007.

[9] G. Foucault, J. C. Cuillierea, V. Francois, and J. C. Leon, “Roland Maranzana. Adaptation of CAD model topology for finite element analysis,” *Computer-Aided Design*, vol. 40, no. 2, pp. 176-196, Febuary 2008.

[10] A. Thakur, A. G. Banerjee, and S. K. Gupta, “A survey of CAD model simplification techniques for physics-based simulation applications,” *Computer-Aided Design*, vol. 41, no. 2, pp. 65-80, February 2009.

[11] J. C. Cuillière, S. Bournival, and V. François, “A mesh-geometry-based solution to mixed-dimensional coupling,” *Computer-Aided Design*, vol. 42, no. 6, pp. 509-522, June 2010.

[12] R. Haimes and A. Merchant, “The synergistic Use of CAD for tightly coupled analysis and design,” in *Proc. 17th AIAA Computational Fluid Dynamics Conference*, Toronto, Canada, 2005, pp. 2005-4986.

[13] A. V. der Velden, “CAD to CAE process automation through iSIGHT-FD,” *Proceedings of the ASME Turbo Expo*, vol. 1, pp. 87–93, 2007.

[14] G. P. Gujarathi and Y. S. Ma, ”Parametric CAD/CAE integration using a common data model,” *Journal of Manufacturing Systems*, vol. 30, pp. 118-132, 2011.

[15] B. Liu, R. T. Haftka, M. A. Akgun, and A. Todoroki, “Permutation genetic algorithm for stacking sequence design of composite laminates,” *Comput. Methods Appl. Mech. Engrg.*, vol. 186, no. 2-4, pp. 357-372, June, 2000.

[16] J. Stegmann and E. Lund, “Discrete material optimization of general composite shell structures,” *Int. J. Numer. Meth. Engng.*, vol. 62, no. 14, pp. 2009-2027, April, 2005.

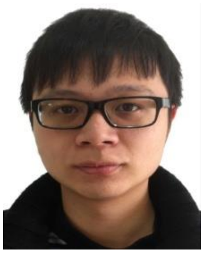
[17] M. Bruyneel, “SFP—A new parameterization based on shape functions for optimal material selection: application to conventional composite plies,” *Struct. Multidisc. Optim.*, vol. 43, no. 1, pp. 17-27, January 2011.

[18] F. X. Irisarri, F. Laurin, F. H. Leroy, and J. F. Maire, “Computational strategy for multiobjective optimization of composite stiffened panels,” *Composite Structures*, vol. 93, no. 3, pp. 1158-1167, February 2011.

[19] M. A. Nik, K. Fayazbakhsh, D. Pasini, and L. Lessard, “Surrogate-based multi-objective optimization of a composite laminate with curvilinear fibers,” *Composite Structures*, vol. 94, no. 8, pp. 2306-2313, July 2012.

[20] V. Premkumar, S. Krishnamurty, J. C. Wileden, and I. R. Grosse, “A semantic knowledge management system for laminated composites,” *Advanced Engineering Informatics*, vol. 28, no. 1, pp. 91-101, January 2014.

Zijun Wu was born on May 23, 1985, in YiChang, China. He obtained his Ph.D. degree in mechanical engineering from Huazhong University of Science and Technology (HUST), China. He now works as a postdoctor researcher in HUST and his research interests include isogeometric analysis and structure optimization. Recently, he has published an article “A local solution approach for adaptive hierarchical refinement in isogeometric analysis” in Computer Methods in Applied Mechanics and Engineering.



Wei Liu was born in Wuhan of China in 1987. He is a Ph.D. student and his major is mechanical design and theory. He obtained his master degree from Huazhong University of Science and Technology, China. His research interests include isogeometric analysis and structure optimization.

Xiaojie Wu was born on July 14, 1989, in Hubei, China. He is a graduate student with a major in mechanical design and theory, in Huazhong University of Science and Technology (HUST), Wuhan, China. His research interests focus on the area of computer-aided design.

Qinghua Liu is an associate professor in School of Mechanical Science and Engineering, Huazhong University of Science and Technology, China. He obtained his first degree in Xi'an Jiaotong University, China, and his Master degree and Ph.D. degree in Huazhong University of Science and Technology. His main research directions are Collaborative Design and Product Lifecycle management.

Zhengdong Huang is a professor in School of Mechanical Science and Engineering, Huazhong University of Science and Technology, China. He obtained his Bachelor degree in Mathematics from Wuhan University in 1983. He received his Master degree in Zhejiang University and Ph.D. degree in Huazhong University of Science and Technology. From 1998 to 2002, he worked as a postdoctoral researcher in University of Michigan. His research efforts focus on the area of CAD/CAM/CAE.