Using Computer-Aided Design to Enhance
Engineering Technology Education

Daniel M. Chen
Central Michigan University

Abstract

This article discusses the use of a computer-aided design (CAD) software as an enhancement to an undergraduate mechanical engineering technology program. With advances in both hardware and software, the technology in CAD is changing rapidly. Although many major software packages in the market are much easier to learn and use, they are more sophisticated in terms of capabilities. It is easy for one to know what the software is doing with visual feedback at every step. This is especially important for engineering technology students who learn best when they can see things and work with them. At Central Michigan University, the CAD software is utilized to enhance students' learning in mechanics of materials, mechanical design and mechanism design. The mechanical/structural problems are used as vehicles to teach proper use of the software and to use it as a way of dealing with various topics incorporated in these subjects. The goal is that the student should be able to grasp the physical problem, understand how a model behaves, check results for accuracy, and know the limitations of the theory on which they are based. In order to investigate the level of success, the instructor often sits with students at the computer to offer advice and monitor their modeling skill and physical understanding of the problem. The focuses of this investigation are on: (1) how the CAD software is used as a visually driven design tool, (2) how the tool allows students to see and increase their understanding the effects of different design parameters, and (3) what difficulties students encounter while using the software.

I. Introduction

Currently in the industry, the CAE software packages have a wide range and sophisticated capabilities. They are becoming more and more user-friendly, easier to operate and master. Some of these software packages in the market today include Unigraphics, CATIA, Pro/ENGINEER, I-DEAS, etc. The capabilities of all these software have evolved greatly and enabled designers to predict the outcome of their designs. Some of these software are currently being utilized by the largest automobile companies. In reference to the goal of reducing the total cost, companies have found that 70% [1] of the total life-cycle cost of a product is determined in the early concept design stage. It is important to make the right choices in this early design phase.

The various courses offered for mechanical engineering technology (MET) program at Central Michigan University have incorporated such a software called I-DEAS, which is developed by Structural Dynamics Research Corporation (SDRC). The Unix-based software employs the concept of 3D master model. As depicted in Figure 1, the 3D master model is a starting point and a shared information source containing the geometric definition of the parts and assemblies

[2]. The 3D master model is used for many "downstream" applications, such as mass properties calculation, mechanism design, FEA, dynamic analysis or other applications.
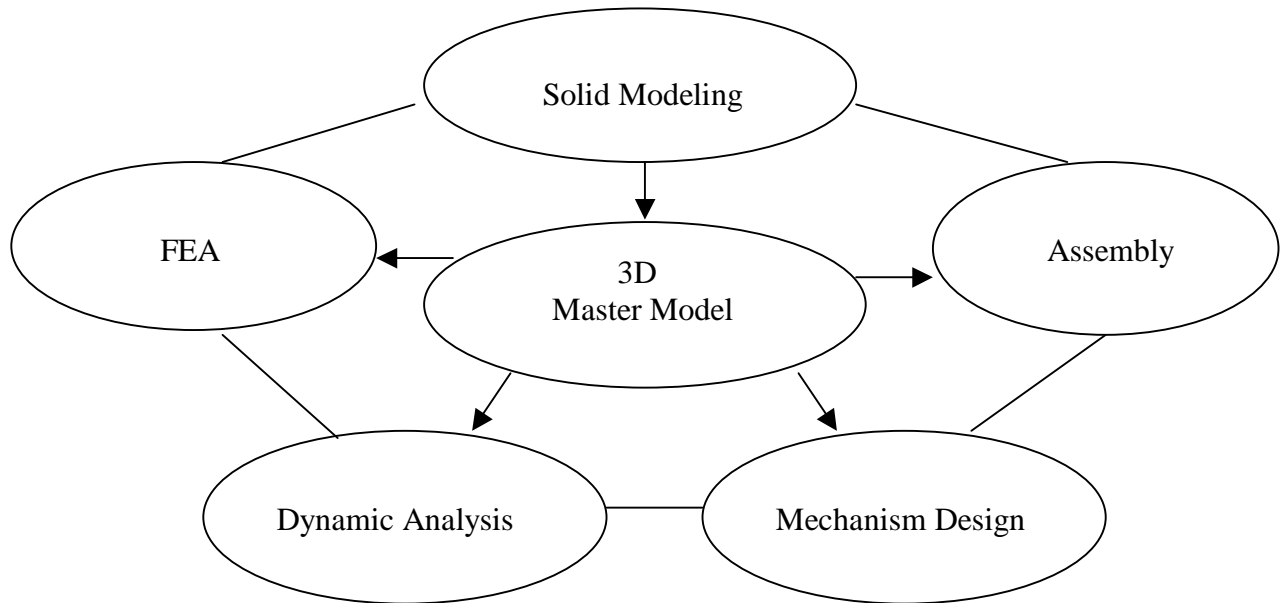

Figure 1. 3D Master Model and Various Applications

Students learn how to use one or the other application of the software during their course of study in MET. Although each of the applications can be used by itself, but the real advantages occur when these tools are used together, allowing them to be used as integrated tools for a design full-cycle. A few year ago, MET program started to offer Advanced Computer-Aided with the idea of concurrent engineering in mind. The senior-level course integrates the different applications into the design phase, resulting in analyzing more concepts in less time. In the sixteen-week semester, students begin with the solid modeling of 3D parts; assemble them for well-defined mechanisms and their motion simulation; and use some of these parts for FEA.

II. Mechanism Design

In this part, students are required to solve the problem with the animation and computation of geometric properties that include displacement, velocity, and acceleration for all the moving parts. Students can show these geometric properties by displaying x-y graphs using the software. One of the assignments in this area is the dumpster mechanism shown in Figure 2. The mechanism must at least consist of a dumpster, a cylinder with a single-end rod, a triangular plate, a connector, and hinges. The following sections discuss how the software is used as visually driven tool for analysis of the mechanism.
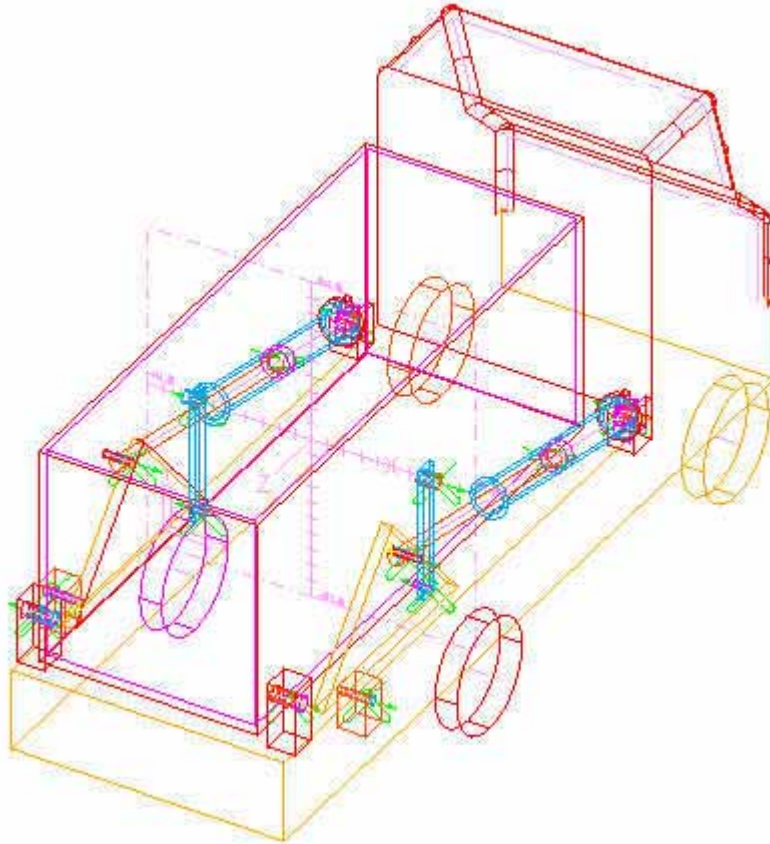
Figure 2. Dumpster Mechanism

<u>Assembly of Solid Parts</u>
Students use solid parts created to define the assemblies and sub-assemblies first. The assembly hierarchy [3] is displayed in a form that allows students to build the assembly according to their relationship with one another. In this case, the title is "dumpster mechanism" which serves as the parent for the hierarchy tree. Students add to this tree the parts and the sub-assembly which is named hinge assembly. Hinge assembly consists of all the hinges. Figure 3 represents the completed hierarchy structure of this mechanism which lists only one of the two symmetric mechanisms analyzed.

Figure 3. Hierarchy Structure

Mechanism Design

In the Mechanism Design task, students need to follow the following steps in order to complete a problem:

(1) apply the proper rigid and ground instances,
(2) apply proper joints to the system,
(3) define motion input to one or more joints,

(4) solve for the motion output of all parts,
(5) animate the motion, and
(6) display the graphs of displacement, velocity, and acceleration for specific parts.

All parts connected with joints must be defined as rigid bodies. The ground instances, in this case, is the sub-assembly. All joints are revolute joints except the translational joints between cylinder and piston. As to the motion input that needs to be assigned to this translational, students are given with the choice of either constant or time dependent.

Mechanism Solution
The result produced from the internal solver of I-DEAS predicts the motion output of all the parts. Before the internal solver is activated, students must define the end time (in seconds) and number of steps in that same period of time. Students usually animate the motion first after a problem has been solved. To complete an animation sequence, the pistons move inward and outward in the cylinders as the dumpster rotates up and down until the number of steps have been completed. In terms of displaying results, students are expected to be able to generate the graphs of linear and angular displacement, velocity, and acceleration of all moving parts, which are plotted as a variable of time. Figure 4 represents the linear acceleration profile of the piston.
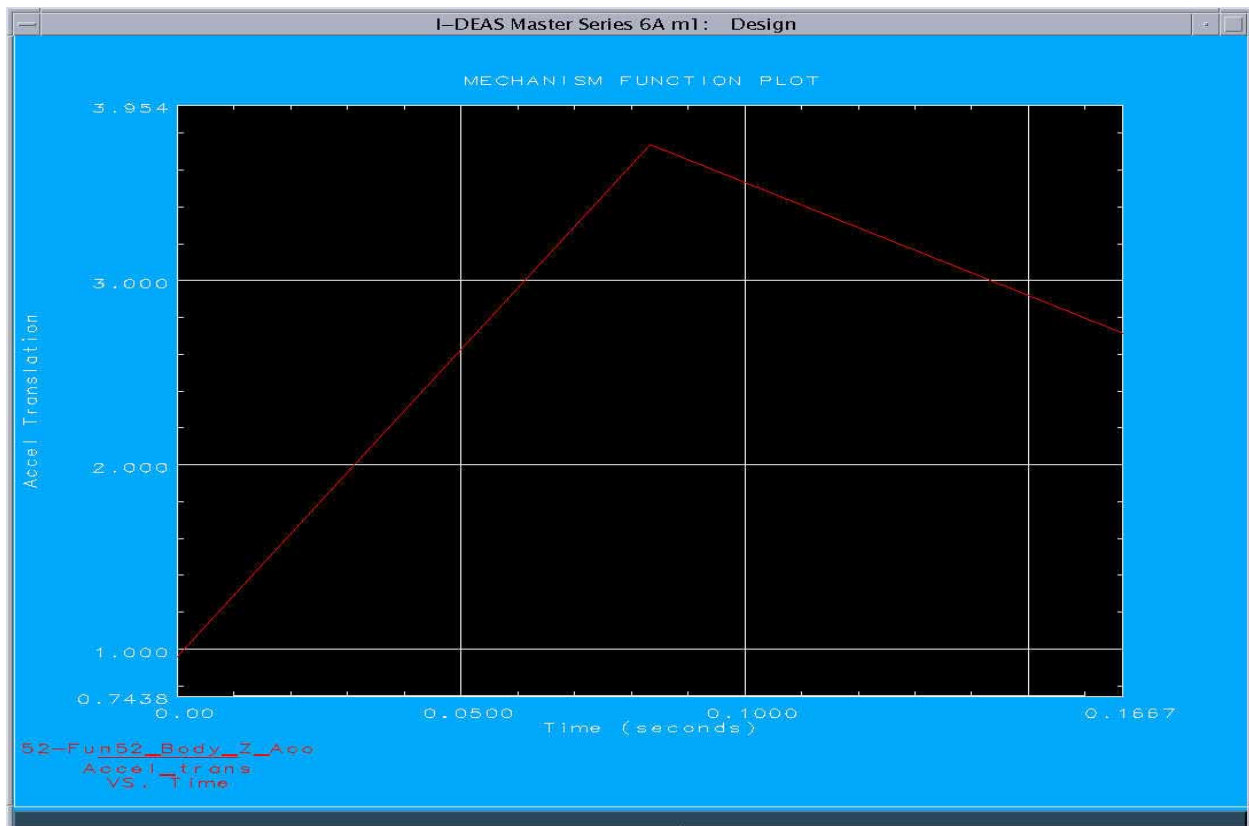

Figure 4. Linear Acceleration Plotted as A Variable of Time

III. FEA

There are four major steps in a FEA process: (1) FE Model Create, (2) Boundary Conditions, (3) Meshing, and (4) Model Solution. Each steps of the FEA process has its own function. The following sections discuss how the software is used as visually driven tool for each of the four steps.

FE Model Create
FEA process begins with FE Model Create. It allows an identified material to be assigned to a part. The material properties, such as modulus of elasticity and Poisson's ratio, can be specified in the material table provided as shown in Figure 5. Students must next determine which method to use for modeling the part. If creating boundary conditions based on geometry is necessary, the option of "Geometry Based Analysis Only" must be selected. If a polynomial function is necessary, the switch of "P Element Analysis" could be turned on to allow the use of a polynomial equation within the element [4]. This means even with fewer larger elements, the convergence is an automatic part of the solution.

Figure 5. Material Properties

Boundary Conditions

The Boundary Conditions task deals with the loads and restraints applied to the part. Students have to be familiar with different types of loads. A load can be either a concentrated force or a distributed pressure. A load can also be a temperature difference or an energy source. Restraints have six values that include three translations and three rotations as depicted in Figure 6. Students must know how to deal with each entry, which can either have a value for the fixed displacement or be left free to move. Figure 6 also shows how both the load (uniform pressure) and the restraints are applied to the half model of the simplified design of a submarine. These well-organized windows bring students into the proper frame of mind to study the physical model, with the goal of being able to deal with each entry correctly.
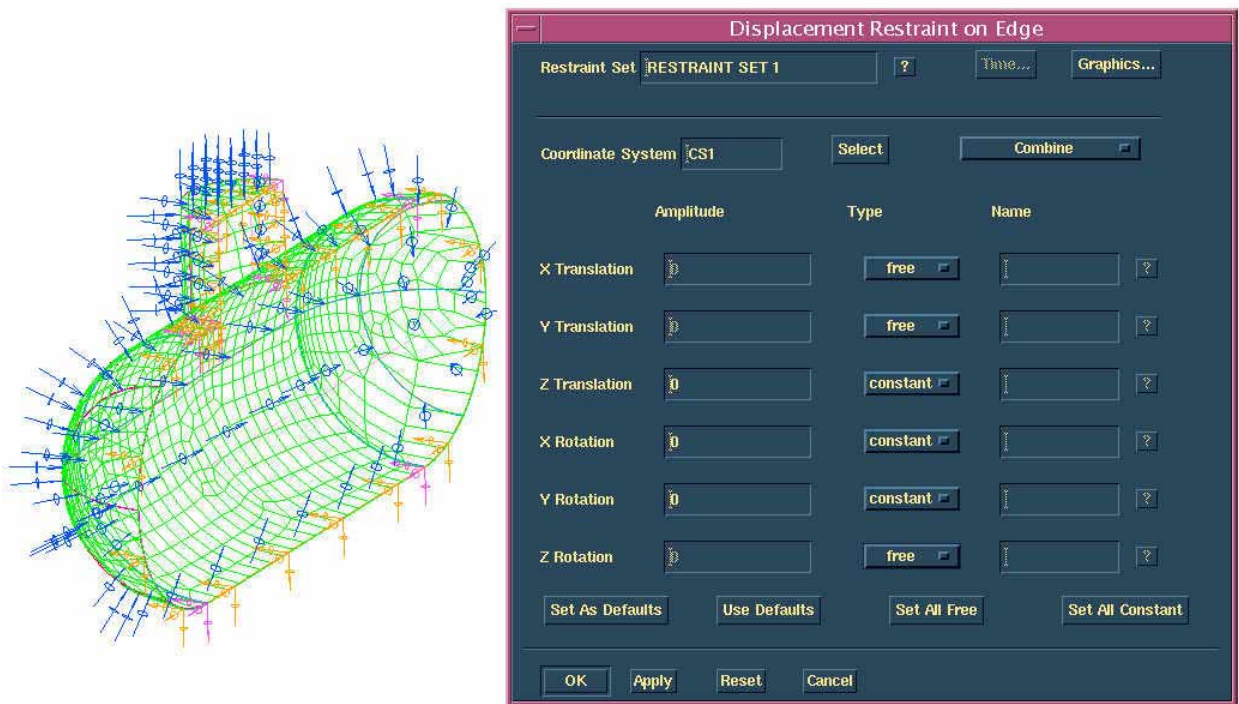

Figure 6. Boundary Conditions and Meshing

Meshing

The Meshing task is used to create nodes and elements and to check the model. Students were instructed what element type to use for a problem – thin shell, solid, or beam-section. Figure 6 shows how the half model of the submarine is meshed with thin-shell elements. Students are evaluated on how effectively they could mesh a model and take the advantage of Quality Check function. Elements are prepared by either free or mapped meshing with the goal to make elements with the minimum distortion and stretch. The objective is to achieve the highest accuracy with the lowest number of elements. Quality Check is a model checking function,

which is used to check the shape of all elements generated from meshing. A lower value from quality check indicates that the elements in the model could be distorted or overly stretched. Students are expected to be able to use this tool to determine whether it is necessary to fix the model by completely re-meshing or just repairing one or a few elements.

Model Solution

The Model Solution task is where the finite element model is solved. It has a number of standard type of analysis that can be performed, which include linear and non-linear statics, linear buckling, heat transfer, and potential flow. This study focuses on linear statics only, because this CAE course serves only undergraduate beginners. Model Solution allows students to select the types of outputs as the results of solution. Stress and deflection are the most common outputs by default. Figure 7 shows the distribution of stress with the color bar on the side for scale. Other outputs like reaction force and element force can be added to the list if necessary.
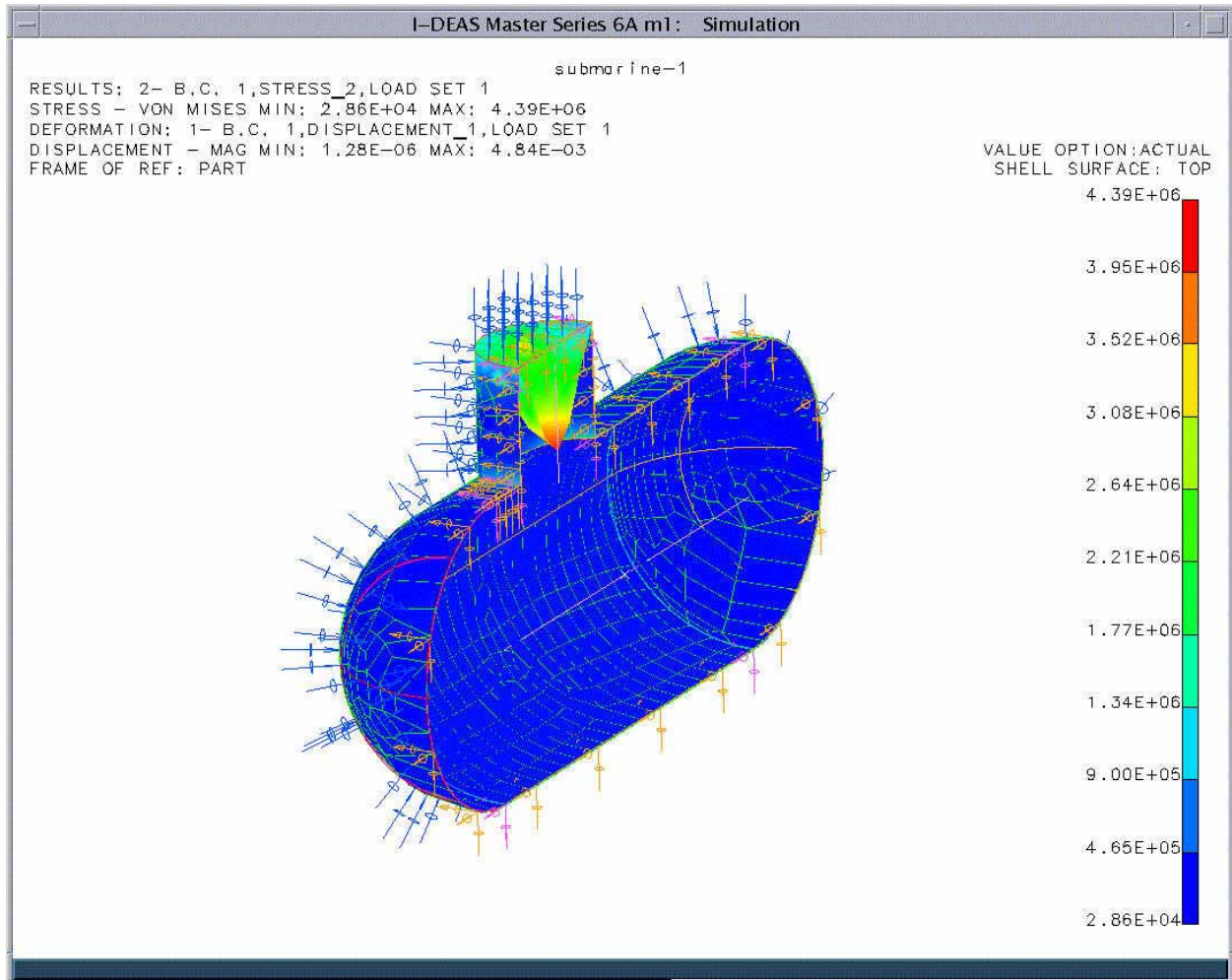

Figure 7. Stress Distribution

IV. Discussion

The software's mechanism design and FEA have been effective enhancement to the students' learning in mechanics of materials, mechanical design and mechanism design. It provides a graphical design tool that helps visualizing the effects of design changes. The details are as follows.

Mechanism Design
According to the instructor's observation, the students' learning process in kinematics/mechanism design is greatly enhanced with the use of the CAE software. Due to the software's capabilities in (1) animation, and (2) display of results, students could use the time saved for more design revisions, which resulted in better understanding of the physical problem. They often generated several trials before finalized a design. The capability of 3D animation would allow configurations be automatically stored for the specific steps of motion. Students would use this function to check the motion of each part relative to the others in a mechanism. Students could also easily check the limit of motion and interference, which were very difficult to be identified with the traditional 2D drafting approach. The graphs of displacement, velocity, and acceleration for all the parts are just a few clicks away once the mechanism has been solved with the software. With the traditional approach, one must plot these graphs manually by determining all three geometric properties one position at a time until the entire cycle was completed. This time-consuming, yet less-accurate, process must be repeated several times for all the moving parts. With use of the software, the laborious process involved in the traditional approach, can be done in a matter of minutes with high accuracy.

FEA
Similar to the mechanism design application, students use the pop-up windows to enter their input data in FEA application. After each run, they can go back to any window, change one or more design parameters and execute another run. The potential of FEA is limited only by the students' physical understanding of a problem.

In general, students had little difficulty dealing with the design of a mechanism using the software. However, this was not the case for FEA. The students' difficulties in FEA are primarily as follows:
1. In Boundary Conditions - if the geometry, restraint, and loading of a FEA model are all symmetric, it is often possible to model only half (or even quarter) of the part. Students encountered this situation quite often, because a large number of FEA problems assigned had parts with the symmetry condition. The difficulty was how to apply the correct restraints to allow the displacements on the symmetry plane of the half model to be identical to that of the entire model. Although the experience from prior coursework helped, it was very common for even a good student to make mistakes in this part of process. This matches the instructor's laboratory observation, that is, very few could correctly handle the restraints required for the symmetry condition at the first trial. However, by dealing with these symmetry conditions, students could understand better on how the models would react to the specific external loads.

2. In Meshing - Students were also expected to be able to use Quality Check to determine whether it was necessary to fix the model by completely re-meshing or just repairing one or a few elements. If a re-meshing was necessary, should it be switched from free meshing to mapped meshing? Very often, the degree of accuracy is determined by how a cut-out feature, such as a hole or a notch, is mapped meshed [5]. The requirement of mapped meshing helped them have a better grasp on predicting stress distribution and be able to carry out re-meshing accordingly.

V. Conclusion

The software has been an eye opener for most students, because it allows the testing of a design without actually building the physical parts. In the course of Advanced Computer-Aided Design, students are encouraged to fully utilize the software in the assigned problems. Most students have been capable of using the software effectively for both mechanism design and FEA applications at the completion of the course. This is particularly important, because they find that by using the software, they have not only acquired a powerful tool that greatly enhances their problem solving skills, but also allows them to understand the concepts of the kinematics and solid mechanics better along the way.

Bibliography
1. *The Fundamentals of I-DEAS Course Guide*, Structural Dynamics Research Corporation, 1996.
2. Sorby, S. A., *Solid Modeling with I-DEAS*, Prentice Hall, 2000.
3. Lawry, M. H., *The I-DEAS Student Guide*, Structural Dynamics Research Corporation, 1997.
4. Adams, V. & Askenazi, A., *Building Better Products with Finite Element Analysis*, OnWord Press, 1999.
5. Shih, R. H., *Introduction to Finite Element Analysis Using I-DEAS Master Series*, SDC Publications, 1999.

DANIEL CHEN
Daniel M. Chen is an Associate Professor of Industrial and Engineering Technology at Central Michigan University. He has taught various courses in Mechanical Engineering Technology during the last fourteen years. Currently, near half of his teaching load is in Computer-Aided Design/Computer-Aided Engineering. Dr. Chen is a registered Professional Engineer in Michigan. He received his Ph.D. in Mechanical Engineering from Kansas State University in 1984. He received his B.S. and M.S. in that same discipline from Taipei Institute of Technology and South Dakota School of Mines and Technology, respectively.