Integration of Finite Element Modeling and Experimental Evaluation in a Freshman Project

Ani Ural[1] and Joseph Yost[2]
[1]Department of Mechanical Engineering, Villanova University, Villanova, PA
[2]Department of Civil and Environmental Engineering, Villanova University, Villanova, PA

Abstract

Engineering analysis, design and research rely on computational and experimental evaluation. In order to prepare undergraduate students for engineering practice and graduate school, it is necessary to build knowledge in both areas throughout the engineering curriculum starting from the first year. The engineering curriculum mostly focuses on laboratory courses that provide exposure to physical testing of mechanical systems. However, the undergraduate experience in computational modeling is limited even though finite element modeling is widely used in current engineering practice. This paper summarizes the integration of finite element modeling with experimental evaluation in a first-year introductory project-based engineering course to provide students with a thorough understanding of current engineering evaluation approaches.

The first component of the project-based course covers the basic theoretical concepts related to the project followed by introduction to finite element modeling and experimental evaluation in a laboratory setting. The initial hands-on experience in finite element modeling allows students to become familiar with general computational modeling procedures as well as to visualize new theoretical concepts such as stress, strain and force-deformation relationships. The computational module is followed by actual beam testing in the structural engineering laboratory where the students have a chance to observe the physical testing and compare the finite element modeling results with experimental data. The integrated approach developed in this project helps students understand how to interface computational and experimental approaches to solve an engineering problem and allows them to evaluate the significance of each component in engineering analysis, design and research. In addition, the exposure of students to finite element modeling at an early stage in their undergraduate education is expected to broaden their understanding of engineering and improve their preparedness for engineering practice after graduation.

1. Introduction

Villanova University has developed a new project-based freshman engineering program to excite students about engineering, to improve the students' understanding of fundamental concepts in engineering and to increase the retention of students beyond the first year.

The new freshman program involves a common freshman year for all freshman engineering students up to the second half of the spring semester when they make their decision on their engineering majors. The first half of the fall semester is a core introduction to engineering course for all freshman students covering basic concepts that relate to all engineering disciplines offered at Villanova University. The core course is followed by multidisciplinary engineering projects which cover the second part of the fall semester and the first half of the spring semester. The students can choose two different projects, one each semester, from the six available projects covering chemical, computer, civil, electrical, and mechanical engineering [1,2]. All projects are multidisciplinary in nature, therefore, have components relating to more than one engineering discipline. This enables the students to have exposure to several areas of engineering before the end of the first year and allow them to make more informed decisions about the engineering discipline they want to continue on during their undergraduate education. Following the multidisciplinary projects, the students declare their majors and take a half semester specialized course in their selected major. The details of the new Villanova freshman program can be found in a previous publication[3].

Analytical and Experimental Evaluation of a SMARTBEAM is one of the new freshman projects. The main goal of the project is to improve the students' understanding of structural behavior via a structural component that is used in engineering practice. The mechanical response of the structural component is evaluated by theoretical, computational and experimental approaches covering all the concepts used in engineering analysis, design and research. The strength of this project comes from its integrated approach in building knowledge about the three main evaluation techniques in engineering during the early stages of the students' undergraduate education. This integrated approach allows the students to get familiar and understand the role of each component in solving a real life problem. In addition, computational methods such as finite element modeling have usually been introduced as a separate course in engineering commonly as a senior level elective despite some efforts to incorporate it in other engineering courses throughout the undergraduate curriculum[4-7]. The use of computational modeling is becoming more common in engineering practice, therefore, the integration of finite element modeling as a complementary tool to various courses throughout the engineering curriculum is expected to enhance the students' understanding of mechanical concepts as well as increase their familiarity with computational modeling. Being exposed to an integrated approach such as in this project is expected to provide the students with a broader perspective in solving engineering problems, to facilitate their critical thinking, and to improve their development as engineers.

In this paper, we will focus on the components of the freshman project, Analytical and Experimental Evaluation of a SMARTBEAM, that are related to understanding of the mechanical behavior of structures. The subsequent sections explain the project structure and the details on each module related to theoretical, computational and experimental modeling.

2. Project Structure

The project combines hands-on in class and laboratory experience with in class lecture style learning. The course was structured as twice weekly, 75-minute-long sessions. The project was taught by three faculty including civil, electrical and mechanical engineering faculty. The components of the project that focused on the mechanical behavior of structures were taught by the authors.

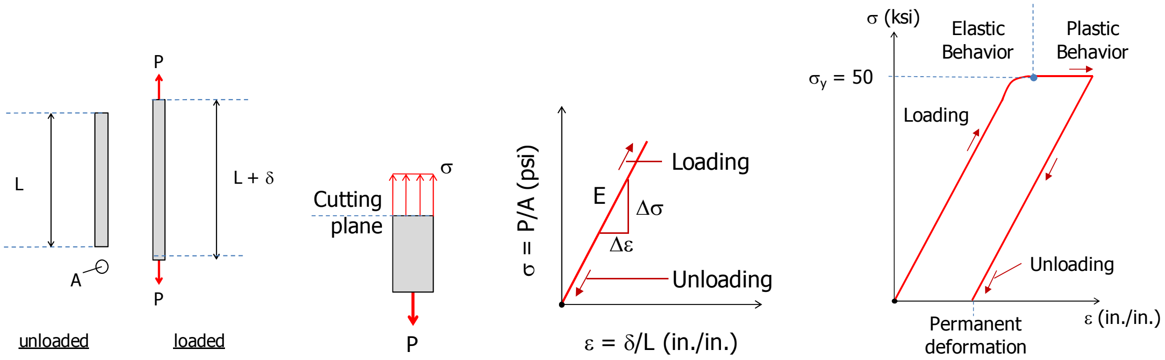
Course related documents including the lecture notes, homework assignments, and tutorials were posted on the course website the day before each class. At the beginning of each class, a clear outline of the lecture and main outcomes of that specific lecture was provided to the students. A short homework assignment was given after each class to improve the students' learning. The homework was done by groups of two students to facilitate teamwork. The students were asked to change their homework partners for each homework.

The project was divided into three modules including theoretical, computational and experimental evaluation. The first component of the project-based course covers the basic theoretical concepts related to the project. The theoretical module was followed by introduction to finite element modeling and experimental evaluation in a laboratory setting. The hands-on experience in finite element modeling allowed students to become familiar with general computational modeling procedures as well as to visualize new theoretical concepts such as stress, strain and force-deformation relationships. The actual beam testing in the structural engineering laboratory provided the students with an opportunity to observe the physical testing and compare the finite element modeling results with experimental data. In the following sections, each project module is explained in detail.

2.1 Basic Theoretical Concepts

The focus of the SMARTBEAM mini project experience is to explore structural behavior using basic theoretical concepts related to axial and flexural loading, the finite element modeling technique, and finally experimentally in the laboratory. To develop basic theoretical concepts students are first introduced to fundamental force-deformation behavior of steel structures using simple thought experiments that are easy to visualize. This exercise begins with simple axial loading in tension. Students are instructed to visualize a simple axial tension test, where a steel rod sample of cross section area A and original length L is loaded in tension with a force P, and then the load is released. Concepts of axial deformation, strain, stress, cutting planes, free body diagrams, and equilibrium are all extensions of this simple exercise. Sample figures related to the discussion are provided in lecture material and shown as well in Fig. 1. Students recognize intuitively that the rod gets longer, the change in length being defined as δ (Fig. 1a). At this point axial strain ε is defined as the change in length divided by original length. Students also recognize the rod is in equilibrium by the forces applied equal in magnitude at each end. An extension of this concept is the cutting plane and exposure of internal material. The force acts along the entire length of the member or is internal along the entire length. This leads to a discussion of normal stress σ, and the understanding that the force is not concentrated at a point in the cross section, but rather all fibers of the section resist the force equally. The force divided by the cross sectional area is defined as normal stress σ, and the distribution is uniform on all material fibers of the cross section (Fig. 1b).

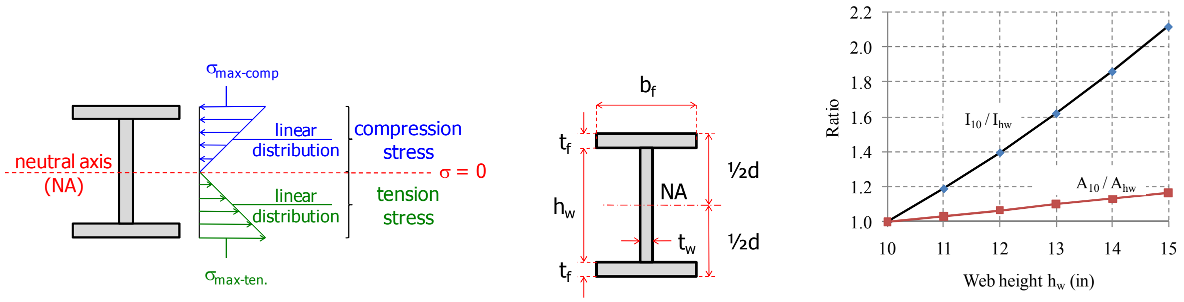
Based on the definitions of stress and strain given above, they are recognized to be normalized forms of force and deformation, respectively. Plotting stress versus strain yields a graph that is characteristic of the material, and not the size and shape of the test specimen. This leads to the development of the stress-strain diagram for steel. As the specimen is loaded the deformation is proportional to the applied force. When the specimen is unloaded the deformation is eliminated (Fig. 1c). This simple understanding introduces the concept of elastic behavior, Young's Modulus E (Fig. 1c) and Hooke's Law $\sigma = E\varepsilon$. The discussion continues and the class recognizes that every structure has finite strength so that at some point the elastic behavior must end. For steel, termination of elastic behavior represents yield of the material and transition to plastic behavior where deformation occurs under constant load (Fig. 1d). When the structure is unloaded in the plastic region there will be permanent or residual deformation. At the conclusion of this simple discussion the students are firmly grounded in force-deformation response and associated elastic and plastic behavior.



(a) (b) (c) (d)

Figure 1: (a) Axial load experiment. (b) Internal material and normal stress. (c) Stress-strain and elastic behavior. (d) Elastic-plastic behavior

The same force-deformation discussion as described above is now developed for bending in beams. Students are instructed to visualize a steel beam in flexure, noting the bending deformation causes material fibers on top to compress and fibers on the bottom to stretch. Thus there is compression on top of the beam and tension on the bottom. Logic dictates that between the top and bottom there must be a location of neutral deformation and neutral stress. This location is defined as the neutral axis (NA) or neutral surface and is shown in Fig. 2a. The bending stress distribution is then recognized to be linear above and below the NA (Fig. 2a) and defined by the flexural formula $\sigma = M(y)/I$. This relationship is first introduced as a function of internal "force" in the numerator, moment in this case, and a property of the cross section in the denominator, moment-of-inertia or I. Thus the bending stress calculation is similar to the axial stress calculation being a function of internal force and a property of the cross section. The cross section is then defined as a series of connected plates, with the top and bottom referred to as flanges with dimensions b_f x t_f, and the vertical middle called a web with dimensions h_w x t_w. The calculation for I is simply given as $\{2x(b_f)(t_f)(½d-½t_f)^2\}_{\text{flange-parts}} + \{{}^1/_{12}(t_w)(h_w)^3\}_{\text{web-part}}$. Using this relationship, a simple parametric investigation is executed where the cross section area (A) and moment of inertia (I) are investigated by increasing h_w between 10 and 15 inches with all other dimensions held constant. The results are shown in Fig. 2c and the exponential growth in I is noted compared with the relatively flat linear increase in A. This (Fig. 2c) is the value of the SMARTBEAM technology, where the beam depth is increased with no net increase in material. Thus, understanding the relationship between stress and moment-of-inertia, and then the dependence of moment-of-inertia on the square of depth, the students can recognize the economy of using a deeper section.



(a) (b) (c)

Figure 2: (a) Bending stress distribution. (b) Member cross section. (c) Parametric study of moment of inertia and cross section area

2.2 Finite Element Modeling

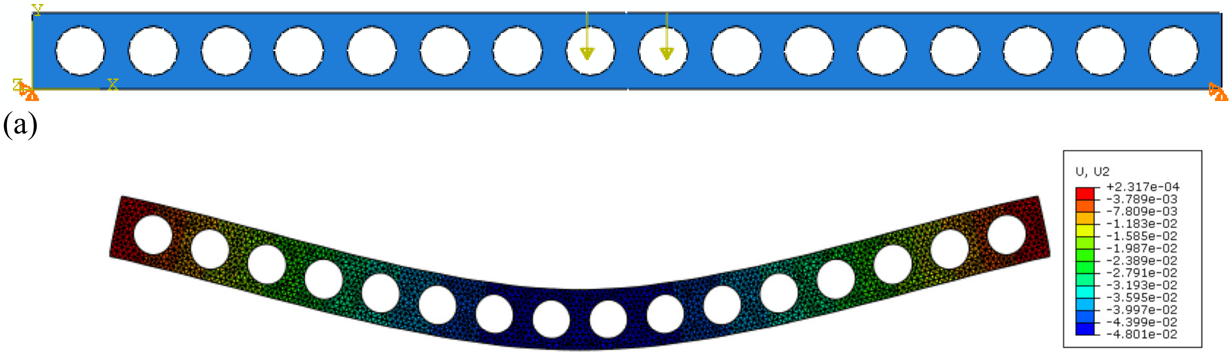
Following the lectures that covered the theoretical concepts about stress-strain relationships, the students were introduced to finite element modeling (FEM). The integration of finite element modeling to the project was aimed to enhance the students' understanding of the mechanics concepts and to introduce students to computational methods and tools during the first year of their undergraduate study. FEM module provided the students with a hands-on learning opportunity where they created the models, ran the simulations, and analyzed the simulation results under the supervision of the instructor in class.

The finite element modeling module of the project was designed with specific learning objectives including (i) understanding the general concept of FEM and its application areas (ii) learning to generate a finite element model of a structure and interpret the results under various loading conditions using a commercial finite element software (iii) gaining a better understanding of mechanical behavior by visualizing stress-strain, load-deformation related concepts in different structures using the finite element software.

In order to address each of the learning objectives the FEM module was divided into three parts. The first lecture was devoted to introduction of general concepts of FEM focusing on case studies from different engineering fields, steps of relating a physical problem to an FEM model, and common pitfalls in FEM analyses. Following the introduction of basic concepts, the specific components of finite element modeling such as generating geometry, defining boundary conditions, applying loads, meshing and identifying material properties were covered through demonstration by the instructor and active participation of students using the finite element program, ABAQUS[8].

The second part of the lecture series on FEM focused on hands-on use of the finite element program and allowed students to apply the new skills that they have learned in the previous lecture to an example problem. The first structure that was chosen for modeling is a two-dimensional square plate with and without a hole under tension. This simple structure was chosen as an initial example for the students to become proficient in main components of FEM including generating geometry, defining boundary conditions, applying loads, meshing, and identifying material properties as well as to develop their understanding of stress distribution, deformed shape, and the concept of stress concentration due to discontinuities in a structure. After building the FEM models and running the simulations, the students investigated the stresses and displacements in both plate models by extracting the values at specific locations. In addition, the students compared the findings from both models and discussed differences and similarities between the mechanical response observed in the two plate models.

The final part of the FEM module was dedicated to the analysis of a cellular SMARTBEAM under four-point bending replicating one of the structural beams to be tested in the experimental module of the project. Using the actual dimensions of the cellular SMARTBEAM, the students built FEM models of the beam (Figure 3a) and performed an FEM analysis to obtain deformation, and stress and strain distribution in the beam under the applied loading (Figure 3b). Specifically, the students were asked to extract the strain and displacement values from the finite element model at specific locations corresponding to points where they will be taking measurements in the laboratory. In addition, the students compared the expected theoretical beam behavior that was covered at the beginning of the project to the finite element model results to improve their understanding of mechanical behavior. At the end of the FEM module, the students have gained proficiency in creating FE models, and have increased their understanding of the mechanical behavior of the SMARTBEAM and have obtained computational results that they can compare to experimental data.



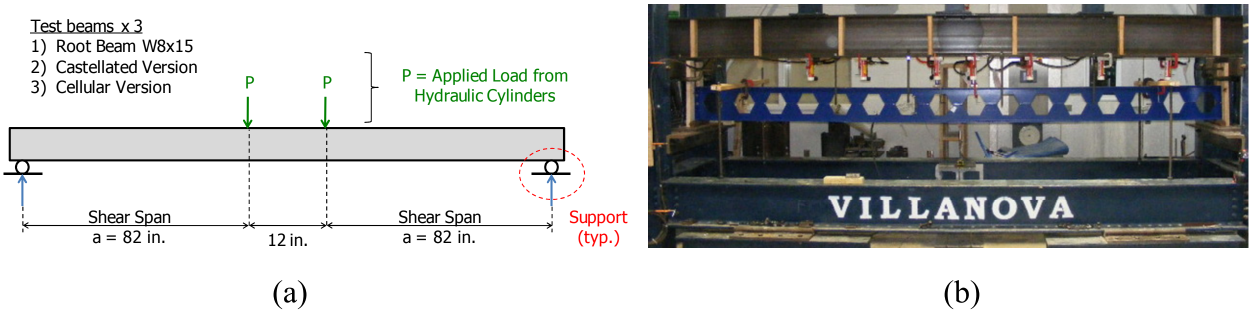
(a)

(b)

Figure 3: (a) Finite element model of the cellular beam. (b) Deformed shape and displacement contour of the cellular beam under four-point bending.

2.3 Experimental Measurements

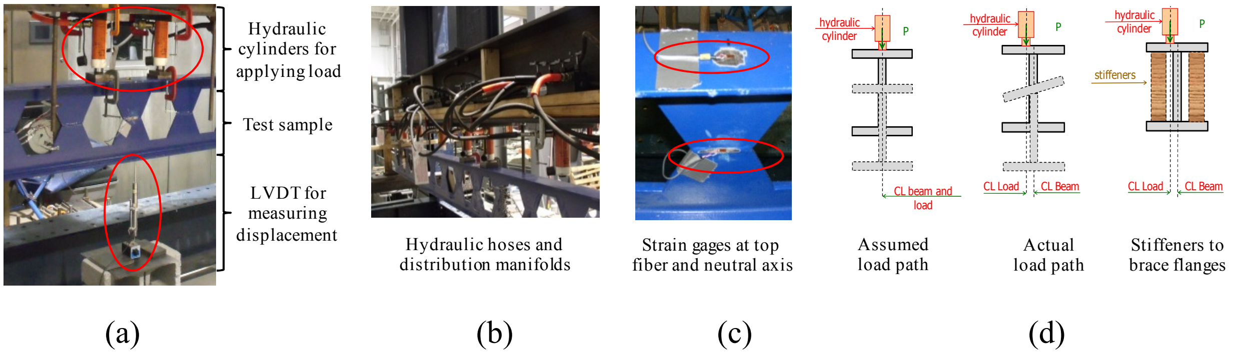
With a firm understanding of basic flexural theory and data from a finite element analysis, the students are introduced to measured experimental behavior of a SMARTBEAM in Villanova's Structural Engineering Teaching and Research Laboratory (SETRL). The test involves four point bending as is shown in Fig. 4. Three individual beams are tested, 1) a root beam to failure, 2) a castellated version of the root beam to failure, and 3) a cellular version of the root beam elastically. Data for the first two tests is provided and the students will enter the SETRL to perform the third test.



(a) (b)

Figure 4: (a) Schematic of test load and support conditions. (b) Photo of SETRL test setup

Before entering the SETRL for the elastic test, students are provided a lecture where the details related to the experimental procedure are described. The intent is to develop a sense of understanding for how the actual test will be executed, as well as to create a sense of anticipation for the event, which they will observe in a subsequent meeting. In describing the testing procedure details related to the following experimental elements are provided: hydraulic cylinders for applying load, hydraulic system components (pump, manifolds, distribution hoses, hydraulic fluid), sample support conditions, strain gage instrumentation, displacement and force transducers, data acquisition system, and finally loading concerns. Elements of this discussion are provided in Fig. 5.



Hydraulic hoses and distribution manifolds

Strain gages at top fiber and neutral axis

Assumed load path

Actual load path

Stiffeners to brace flanges

(a) (b) (c) (d)

Figure 5: (a) Load and displacement. (b) Hydraulic system. (c) Stability at load point

Data plots in the form of load-deflection (Fig. 6a) and load-strain (Fig. 6b) for the root beam (W8x15) and castellated beam are provided and discussed before visiting the SETRL. The students perform hands-on elastic testing on the cellular beam in SETRL and observe the mechanical behavior via a physical test. At the end of the experimental module, students are able to correlate elastic and plastic behavior and the stress distribution as related to maximum fiber stress at top and bottom, and zero stress at the middle fiber.

Finally, the experimental results are compared to the expected behavior predicted using the basic theoretical flexural relationships and results from the finite element model. Importantly the students are exposed to the importance of comparing predicted behavior with measured behavior and that this comparison allows engineers to objectively understand the value and accuracy of their mathematical model. The correlation is, for simple systems with known load, support and material conditions, usually quite good. However, the analysis of large complex structures often requires simplifying assumptions, and engineers must rely on measured behavior to know if their model represents realistically well the behavior of the physical system.

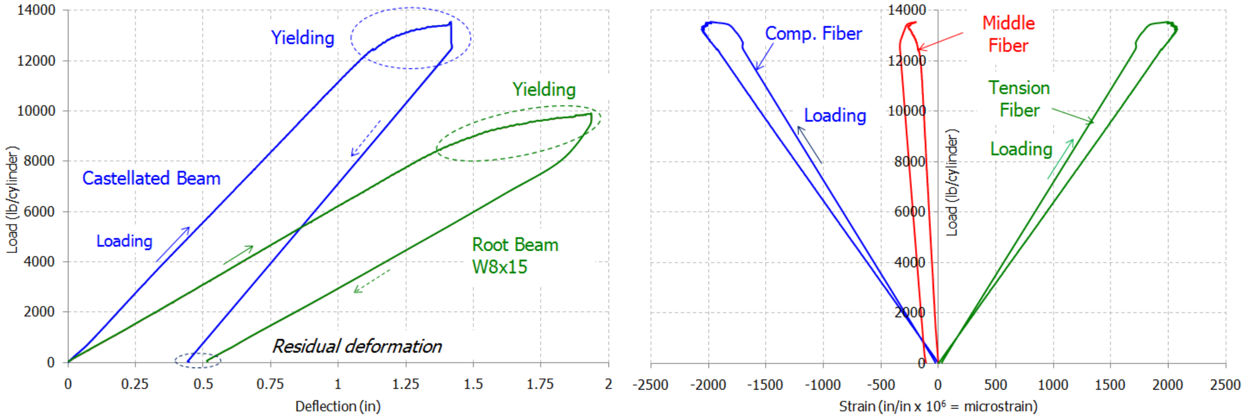


Figure 6: (a) Load-deflection results. (b) Load-strain results

2.4 Integration of theoretical concepts, finite element modeling and experimental measurements

At the conclusion of the project the students have observed the mechanical behavior of the SMARTBEAM in the laboratory and by using computational modeling, therefore, they have been exposed to different approaches to analyzing the same problem. In order to improve the students' understanding of theoretical, computational and experimental concepts and to facilitate the integration of these concepts further, the students are asked to write a technical report that summarizes each component of the project and relates each component to the overall understanding of mechanical behavior. Combining the components at the end of the class as a technical report is expected to increase the understanding of the concepts learned during the semester. The evaluation of the finite element results and experimental findings provide an opportunity for the students to understand the limitations and sources of error in computational and experimental approaches. Furthermore, the integration of all components through a technical report allows students to reflect on the interrelation between computational and experimental components and their respective significance in engineering analysis, design and research.

3. Discussion and Conclusion

As a part of the new Villanova freshman engineering program, the project “Analytical and Experimental Evaluation of a SMARTBEAM”, provides an integrated approach to engineering education by combining theoretical, computational and experimental components. The engineering curriculum mostly focuses on laboratory courses that provide exposure to physical testing of mechanical systems. However, the undergraduate experience in computational modeling is limited even though finite element modeling is widely used in current engineering practice. The integrated approach such as the one developed for this project provides a platform similar to engineering design and analysis in practice and present a thorough understanding of current engineering evaluation approaches.

Integration of finite element modeling with experimental investigations improves the students' understanding of the experimental observations made in the laboratory setting and enhances their mechanics understanding. Implementation of new technologies such as finite element modeling in a freshman level project-based course not only provides a powerful visualization tool for mechanical concepts that are new to the freshman level students but also increases the students' interest in engineering. In addition, the integrated approach developed in this project helps students understand how to interface computational and experimental approaches to solve an engineering problem and allows them to evaluate the significance of each component in engineering analysis, design and research. The exposure of students to finite element modeling at an early stage in their undergraduate education is expected to broaden their understanding of engineering and improve their preparedness for engineering practice after graduation.

References

[1] Dinehart, D., Yost, J., Gross, S., Radlinska, A. T*he role of structural engineering in multi-disciplinary freshman projects*. American Society for Engineering Education Annual Conference & Exposition, 2010.

[2] Comolli, N., Kelly, W., Wu, Q. *The artificial kidney: investigating current dialysis methods as a freshman design project*. American Society for Engineering Education Annual Conference & Exposition, 2010.

[3] Caverly, R., Fulmer, H., Santhanam, S., Singh, P., O'Brien, J., Jones, G., Char, E., Mercede, F., Weinstein, R., Yost, J. *Project-Based Freshman Engineering Experience: The Core Course* American Society for Engineering Education Annual Conference & Exposition, 2010.

[4] H. Jenkins and J. Mahaney, *Improving mechanical design education using solid modeling and finite element analysis modules*, ASEE Southeastern Section Conference, 2005.

[5] L. C. Brinson, T. Belytschko, B. Moran and T. Black, *Design and computational methods in basic mechanics courses*, Journal of Engineering Education 86 (1997), 159-166.

[6] W. E. Howard, J. C. Musto and V. Prantil, *Finite element analysis in a mechanics course sequence*, American Society for Engineering Education Annual Conference & Exposition, 2001.

[7] C. J. Lissenden, G. S. Wagle and N. J. Salamon, *Applications of finite element analysis for undergraduates*, American Society for Engineering Education Annual Conference & Exposition, 2002.

[8] ABAQUS, Teaching Edition, version 6.9, 2009, Simulia , Providence, RI