AC 2007-1938: INTRODUCING FINITE ELEMENT ANALYSIS IN THE FIRST COURSE OF STATICS AND SOLID MECHANICS

Pramod Chaphalkar, Grand Valley State University

Dr. Chaphalkar received his M.S. degrees in Mechanical Engineering from Indian Institute of Technology, Mumbai (Bombay), and Ph.D. degree in Mechanical Engineering from North Carolina A&T State University. He has worked in Bajaj Auto, India and General Motors, US. He has industrial experience in the analysis and design of various vehicle components; testing and experiments; finite element modeling and analysis; development of engineering software; and training; vehicle crash simulations; interface with suppliers, consultants and universities. Dr. Chaphalkar's research and teaching interests include Solid Mechanics, Finite Element Analysis, Automotive Crash, Product Design, and Composite Materials.

David Blekhman, California State University Los Angeles

Introducing Finite Element Analysis in the First Course of Statics and Solid Mechanics

Abstract

To remain competitive companies are increasingly adopting a so called *math-based* strategy of virtual product development wherein Computer Aided Engineering (CAE) tools, including Finite Element Analysis (FEA) are used. Teaching FEA has thus become a necessity for the today's undergraduate mechanical engineering programs. At Grand Valley State University (GVSU), we strive to keep our curriculum up to date, reflecting the demands of industry. We have therefore begun the process of integrating the use of FEA tools throughout the curriculum, instead of delaying it until the senior year either for senior design or elective courses. This paper describes the introduction of FEA to students in the first course of Statics and Solid Mechanics. The first priority of this course is to build the foundation for Mechanics. The challenge therefore was to determine the content without compromising the priority. Keeping this in mind, 1-D Bar elements and 2-D Truss elements are introduced in the course. These two limited topics are chosen because they are based solely on the fundamentals of the first course of Statics and Solid Mechanics. The 1-D Bar element is taught as an extension of an axially loaded member. The stiffness matrix for this element is derived using the force equilibrium at a node. Later the concept of the 1-D Bar element is extended to the 2-D Truss element with trigonometric transformation and matrix manipulations. Essentially it is a matrix method of structures with an FEA flavor. Advantage of this methodology is that the students can perform more complex analysis such as a truss design project. With FEA software tools such as ANSYS, they can quickly make design changes and complete more design iterations. They are required to analyze the final iteration using MATLAB or SCILAB. At the same time, they are also required to perform hand calculations, thus providing experience in validating the results produced by an FEA package. The inclusion of this introduction to FEA reinforces their understanding of the concepts of Statics and Solid Mechanics for the cases of axially loaded member and truss structures with increased capability of analysis. Throughout the course, the importance of understanding fundamental FEA techniques is emphasized for becoming a "well educated" engineer, as opposed to mere "well trained" user of FEA tools.

Introduction

The highly competitive business environment requires that the product development processes need to be faster and less expensive. To remain competitive companies are increasingly adopting a so called *math-based* strategy of virtual product development wherein FEA tools are used. FEA needs no introduction. With the advent of cheap computing power and proliferation of commercial softwares with strong capabilities to model complex geometries, FEA has become the *de-facto* method in stress analysis. It is also widely used in others areas such as vibrations, heat transfer, and fluid flow. With faster turn around rates, i.e. time required from meshing to final results, FEA is an integral part of the modern product development. To keep up with this development, teaching FEA has become a necessity for today's undergraduate mechanical engineering programs. Also, one of the ABET criteria is familiarity with modern engineering tools, and FEA certainly fulfils that criterion.

The introduction of finite element analysis (FEA) in undergraduate curriculum is not new. Traditionally many universities have an FEA course as a technical elective in senior level[1,2]. Almost all instructors use FEA for mechanical design projects[3]. Some instructors have developed case studies so that FEA software can be incorporated into engineering courses[4]. Many educators introduce FEA in lower level mechanical engineering courses, most likely in Mechanics of Materials[5, 6, 7]. Some found it beneficial for the mechanical engineering students to have exposure to FEA as frequently as possible in their engineering education [8]. Some educators even found than FEA can be introduced to students at an even earlier point in the curriculum, i.e. Statics by introducing first the deformation theory [9] or without deformation theory [10].

Obviously the approach of introducing FEA has been used in both a statics and a mechanics of materials class. However, there is a wide diversity of university backgrounds, course sequence and structure, students' and local industry's needs, pedagogical objectives and approaches, etc. This paper describes the introduction of FEA in the first course of statics and solid mechanics in our specific context.

Background

Grand Valley State University (GVSU) is located in the west part of Michigan and is surrounded by many auto suppliers and furniture manufacturers. Our engineering school has strong partnership with the local industry and we strive to keep our curriculum up to date, reflecting their needs. All of our senior design projects are sponsored. Co-op is mandatory in our undergraduate program. When our students work in the industry as a part of the co-op program, they are exposed to the FEA tools. We are concerned that students tend to become fascinated with colorful outputs of FEA and neglect to understand the underlying fundamental principles, with their strengths and limitations. Many employers also indicated that they wanted students to be familiar with the FEA tools. We have therefore begun the process of integrating the use of FEA tools throughout the curriculum, instead of delaying it until the senior year, either for senior design or elective courses. A typical statics and solid mechanics related course sequence with co-ops is as follows:

Fall (Third Semester):	EGR209 Statics and Solid Mechanics
Summer:	EGR290 Engineering Co-op I
Winter:	EGR390 Engineering Co-op II
Summer (Sixth Semester):	EGR309 Machine Design I (a solid mechanics course)
Fall:	EGR490 Engineering Co-op III
Winter (Seventh Semester):	EGR409 Machine Design II
	EGR485 Senior Project I
Summer (Eighth Semester):	EGR486 Senior Project II

This paper describes the introduction of FEA to students in the first course of Statics and Solid Mechanics (EGR209). The statics part of this course contains traditional topics such as rigid body equilibrium, frames and trusses. The solid mechanics part of the course contains introductory topics such as stress and strain due to axial and torsional loading, statically indeterminate axially and torsionally loaded members, etc. The first priority of this course is to build the foundation for Mechanics. This course does not have a lab component. The challenge, therefore, was to determine the content in this context. Keeping this in mind, a 1-D Bar element and a 2-D Truss element are included in the course. The 1-D bar element is taught as an extension of an axially loaded member. Later the concept of the 1-D Bar element is extended to the 2-D Truss element with trigonometric transformation and matrix manipulations. The objectives and approach of integration of FEA are described below.

Objectives

- Teach the basic steps in finite element methods (FEM): meshing, element equations (element stiffness matrix), assembly of element stiffness matrices, application of boundary conditions and solution.
- Develop these FEM concepts from elementary statics and solid mechanics.
- Familiarity with commercial software such as ANSYS.
- Not to use just an FEA software but also use additional software like to MATLAB along with traditional statics hand calculations.
- Critique and interpret the FE results and thereby reinforce the methods of statics.
- Employ the above mentioned FEA skills to a real life-like design problem.
- Make the students "well educated" engineers, as opposed to mere "well trained" users of the FEA tools.

Structure of the FEA Instructions

Introduction of FEA material required about five contact hours. This time was carved out by dropping such topics as shear and bending diagrams and mass moments of inertia, which are addressed in later courses. The table 1 shows how the FEA was integrated in the traditional course by describing what part was taught in the class room and what part that the students learned out side the class room. It also shows that the final course project using ANSYS, MATLAB and hand calculations was an important part of the course. What is new here is the totality of all the key components (shown in the Table 1) such as in-class discussion, out-of-the class self-learning, final project and context of our co-op program.

Supplementary textbook style full length chapters and tutorials are written and are provided to the students throughout the course. There are only few FEA textbooks (such as by Cook[11]) that can be used in traditional undergraduate mechanics courses. The chapters and tutorials can not be provided here because of the obvious space limitations but the interested readers can approach the authors to discuss the same.

Table 1

	Chapters / Topics	Mode of Instruction	Deliverables
1	Introduction to FEA	Outside the classroom self-reading material for general introduction.	This part is not tested.
2	Introduction to matrix algebra with MATLAB tutorial	Outside the classroom self-reading material. Matrix methods are taught concurrently in a mathematics course which is a co-requirement for this course.	Home work problems
3	Analysis of axially loaded structures with 1-D Bar Elements	In-class material. Class room presentation and discussion	Home work problems using MATLAB
4	Analysis of planar trusses with 2-D Truss Elements	In-class material. Class room presentation and discussion	Home work problems using MATLAB
	Both of these above mentioned chapters contain solved example problems and the end-of-the-chapter home problems with solutions manual available to the instructors.		
5	Introduction to ANSYS: A Tutorial on 2-D Trusses	Outside the classroom self-reading material. Students follow the same problem which was discussed in the class room.	Home work problems using ANSYS
6	Truss design project: One of the final course project in which students come up with a design with minimum material cost. With ANSYS, they can quickly make design changes and complete more design iterations.	Outside the classroom activity	They are required to analyze the final iteration using MATLAB and to perform hand calculations, using method of joints and sections.

Theory

As discussed in the Background section, this is a first course of Statics and Solid Mechanics. With a limited topical background only direct stiffness method can be used for the 1-D bar element and the 2-D truss element. Hence it is indeed a matrix analysis approach presented with FEA flavor. However, this course is more than mere 1-D bar elements and 2-D truss elements as discussed in previous section.

Axially Loaded Structures and 1-D Bar Element

The 1-D bar element was explained by comparing end forces and deflection with those of a bar under axial load (Figure 1).

Figure 1 Comparison of an axially loaded bar and a 1-D Bar element

Comparing the end forces

$$f_i = -F = -k\,\delta = -k\,(u_j - u_i) = k\,u_i - k\,u_j$$

$$f_j = F = k\,\delta = k\,(u_j - u_i) = -k\,u_i + k\,u_j$$

From these equations the concept of element equations (and the element stiffness matrix) naturally follows as:

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i \\ f_j \end{Bmatrix}$$

Equations of the entire system (or assembly of the global stiffness matrix) were explained via an example of two elements (Figure 2).

For element 1 $\qquad\qquad$ For element 2

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \end{Bmatrix} \quad \text{and} \quad \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix}$$

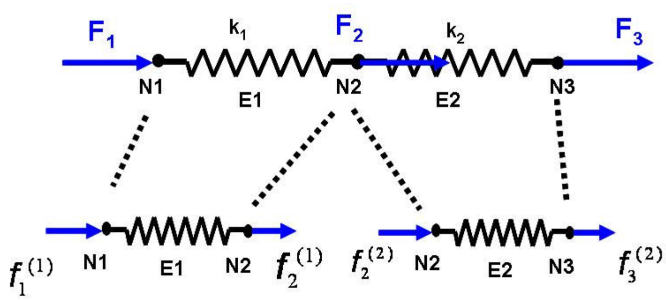

Figure 2 Assembly of two elements

The force equilibrium at every node leads to

$$F_1 = f_1^{(1)} = k_1\,u_1 - k_1\,u_2$$
$$F_2 = f_2^{(1)} + f_2^{(2)} = -k_1\,u_1 + k_1\,u_2 + k_2\,u_2 - k_2\,u_3$$
$$F_3 = f_3^{(2)} = -k_2\,u_2 + k_2\,u_3$$

From these equations the concept of system equations (and the global stiffness matrix) naturally follows as:

$$
\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}
$$

The assembly of the global stiffness matrix is taught by recognizing the pattern (Figure 3a) in the above matrix equation. Recognition of the pattern leads to the general assembly procedure (Figure 3b).

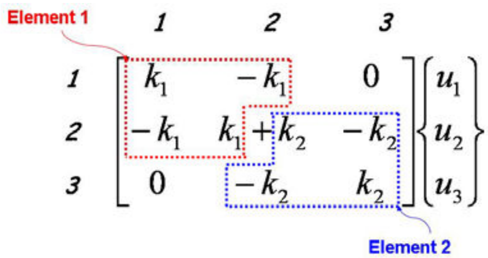

Figure 3a. Global stiffness matrix for two-element assembly

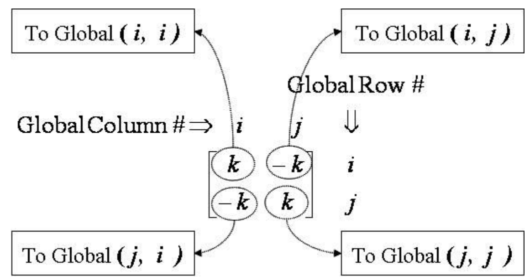

Figure 3b. Assembly process from the element stiffness matrix

The boundary conditions were explained by both methods, namely unconstrained structures and singular global stiffness matrices. They were incorporated using the penalty method or very stiff springs at the supports (Figure 4)

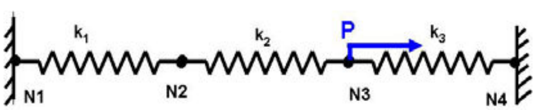

Figure 4a. Example for boundary conditions

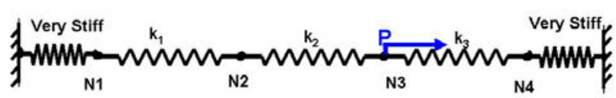

Figure 4b. Very stiff springs as supports

Solution techniques were not covered as matrix methods are taught concurrently in a mathematics course which is a co-requirement for this course. Solutions were obtained simply using MATLAB.

2-D Trusses

The stiffness matrix of the 2-D Truss element was treated as an extension of the 1-D Bar element in local coordinates (Figure 5).

$$
\begin{bmatrix} k & 0 & -k & 0 \\ 0 & 0 & 0 & 0 \\ -k & 0 & k & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_{ix} \\ u_{iy} \\ u_{jx} \\ u_{jy} \end{Bmatrix} = \begin{Bmatrix} f_{ix} \\ f_{iy} \\ f_{jx} \\ f_{jy} \end{Bmatrix}
$$

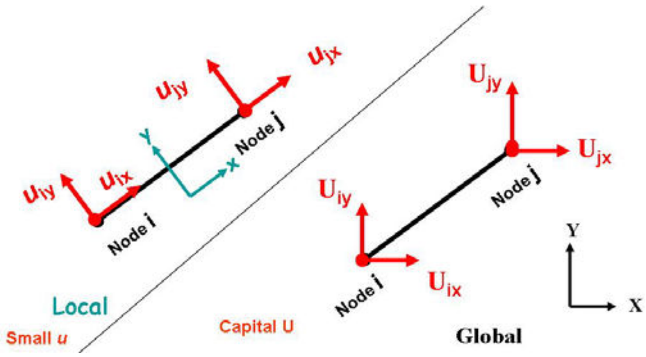

Figure 5. 2-D Truss element as 1-D Bar element

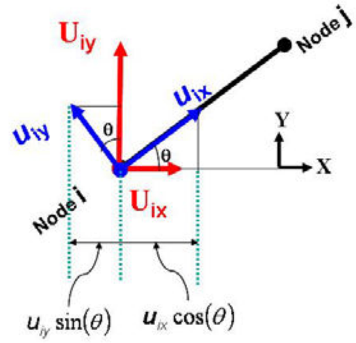

Figure 6. Trigonometric transformations

The stiffness matrix in the global coordinates is obtained with trigonometric transformations (Figure 6)

$$\{\mathbf{U}\} = [\mathbf{T}]\{\mathbf{u}\} \quad \text{and} \quad \{\mathbf{F}\} = [\mathbf{T}]\{\mathbf{f}\}$$

where the transformation matrix [**T**] can be shown to be

$$[\mathbf{T}] = \begin{bmatrix} \cos\theta & -\sin\theta & 0 & 0 \\ \sin\theta & \cos\theta & 0 & 0 \\ 0 & 0 & \cos\theta & -\sin\theta \\ 0 & 0 & \sin\theta & \cos\theta \end{bmatrix}$$ with an interesting property $[\mathbf{T}]^{-1} = [\mathbf{T}]^{T}$

Manipulating these matrices using $\{\mathbf{u}\} = [\mathbf{T}]^{-1}\{\mathbf{U}\}$, $\{\mathbf{f}\} = [\mathbf{T}]^{-1}\{\mathbf{F}\}$ leads to

$$[\mathbf{k}]^{(e)} = [\mathbf{T}]\,[\mathbf{k}]\,[\mathbf{T}]^{T}$$

which finally yields

$$[\mathbf{k}]^{(e)} = k\begin{bmatrix} c^2 & c\,s & -c^2 & -c\,s \\ c\,s & s^2 & -c\,s & -s^2 \\ -c^2 & -c\,s & c^2 & c\,s \\ -c\,s & -s^2 & c\,s & s^2 \end{bmatrix} \quad \text{with } k = \frac{AE}{L} \quad c = \cos\theta \quad \text{and } s = \sin\theta$$

The assembly procedure and the incorporating boundary conditions were treated in a similar fashion that was used in axially loaded structure.

Critique and Interpretation of the Results

Finite element methods (FEM) for axially loaded structures and trusses are *exact* within the respective theories. Hence the numerical or the approximate nature of FEM was explained to the students but it could not be demonstrated in these cases. However, user mistakes such as wrong dimensions, material properties, loading and boundary conditions were emphasized as FEA software may produce the results surreptitiously with erroneous inputs. For all the problems and the course project the students were required to verify their results by

1. Checking the reactions forces.

2. Passing an arbitrary section through the structure and making sure that the element internal force and other applied forces are in equilibrium (Figure 6).

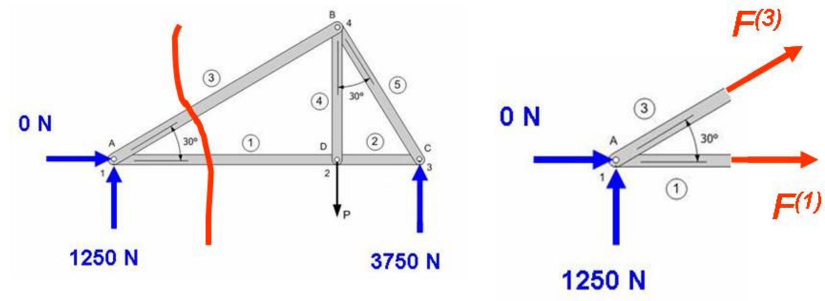

Figure 6. Verification by section method and equilibrium

3. Making sure that the sum of the forces at any node should be zero (Figure 7).

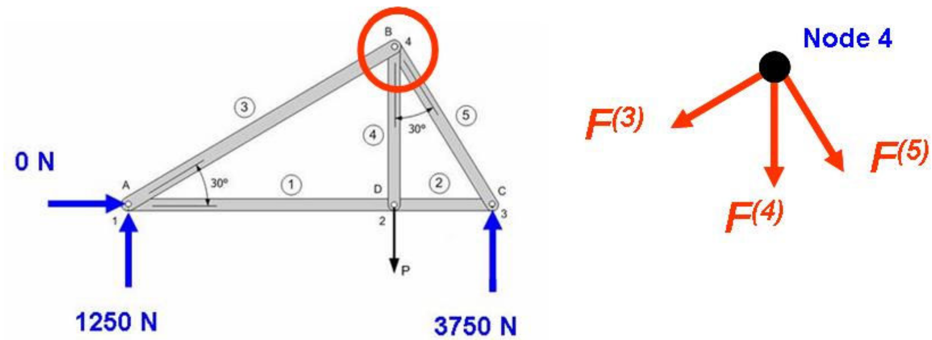

Figure 7. Verification by node equilibrium

Outcomes & Conclusions

Students learned the basic steps in FEM: meshing, element stiffness matrix, assembly of the global stiffness matrix, application of boundary conditions and solution; for axially loaded and planar truss structures. They used matrix methods with MATLAB, thus getting a glimpse into the black box of commercial FEA software.

The FEA lecture material was supported through homework assignments and a bridge design project. The homework for 1D elements consisted of two statically determinate and one indeterminate problem; all of them were completed employing MATLAB. The homework for 2D elements consisted of two truss problems. A seven-member truss was solved first using MATLAB and second ANSYS, while a 30-plus-member truss was solved purely with ANSYS preceded by a demonstration in class and a custom tutorial. The latter did not contain any elements of the assigned problem and merely provided a description of the steps and commands appropriate for such an assignment

Students' evaluations indicated that they were overwhelmed while solving the truss problems using MATLAB interactively. Some students wrote functions to generate element stiffness matrices, to assemble the global stiffness matrix and to transform global solution into local coordinates for stress calculations. The challenge was well received.

Students learned one commercial FEA software, ANSYS. They were particularly enthusiastic in learning ANSYS as they would look forward to use FEA in their co-ops. The bridge design project required students to design a bridge to given specifications. Type of material, number of members and joints, over-under the deck, and member profiles were among several open ended options in the design with the overall goal of building the cheapest bridge to handle the prescribed loading. Students evaluated designs using ANSYS progressively improving with each iteration. Due to the need to account for the tensile and buckling failure modes as well as widely varying load magnitudes, cross-sectional areas of the truss members were made solid or hollow and of various dimensions. Students were comfortable with selecting and properly applying elements with different properties to the design. Final designs reflected their substantial understanding of the related topics in statics, solid mechanics, and FEA.

For all the problems and the course project the students were required to verify FEA, including ANSYS, results as mentioned in the above section. This component gave them additional practice in free body diagram and equilibrium equations and made them "well educated" engineers, as opposed to mere "well trained" users of the FEA tools. Also the inclusion of FEA in earlier course of statics and mechanics complements the traditional analysis techniques.

Future Developments

We are in the process of compiling and editing the above mentioned chapters and tutorials to maintain flow and consistence and to form a single course pack.
We need to fine tune some small details which are not mentioned in this paper for the sake of brevity. MATLAB teaching needs to be more appropriate and effective for the FEA.
The integration of FEA will be further implemented in the next course in the Solid Mechanics (i.e. EGR309 and EGR409) where the applications of FEA to bending of beams and plane stress cases are covered.

References

1. Howard, W. E., Musto, J. C., and Prantil, V., “Finite Element Analysis in a Mechanics Course Sequence,” Proceedings of the 2001 American Society for Engineering Education Annual Conference & Exposition, Session 2793, 2001.
2. Lissenden, C. J., Wagle, G. S., and Salamon, N. J., “Applications of Finite Element Analysis for Undergraduates,” Proceedings of the 2002 American Society for Engineering Education Annual Conference & Exposition, Session 3568, 2002.
3. Burkhardt, J., “Design, Build and Test in Support of Computer Aided Design”, Proceedings of the 2006 American Society for Engineering Education Annual Conference & Exposition, Session 1455, 2006.
4. Ryan, R. G., Prince, S. P., “Development of Engineering Case Studies for Integrating Finite Element Analysis into a Mechanical Engineering Curriculum”, Proceedings of the 2005 American Society for Engineering Education Annual Conference & Exposition, Session , 2005.
5. Mueller, D. W., Jr., “Introducing the Finite Element Method to Mechanical Engineering Students Using Matlab,” Proceedings of the 2003 American Society for Engineering Education Annual Conference & Exposition, Session 3566, 2003.

6. Jolley, W. O., Rencis, J. J., and Grandin, H. T., Jr., “A Module for Teaching Fundamentals of Finite Element Theory and Practice Using Elementary Mechanics of Materials,” Proceedings of the 2003 American Society for Engineering Education Annual Conference & Exposition, Session 3268, 2003.
7. Coyle, M. F. and Keel, C. G., “Finite Element Analysis to Second Year Students,” Proceedings of the 2001 American Society for Engineering Education Annual Conference & Exposition, Session 2793, 2001.
8. Baker, J. R., Capece, V. R., and Lee, R. J., “Integration of Finite Element Software in Undergraduate Engineering Courses,” Proceedings of the 2001 American Society for Engineering Education Annual Conference & Exposition, Session 1520, 2001.
9. Pike, M., “Introducing Finite Element Analysis in Statics,” Proceedings of the 2001 American Society for Engineering Education Annual Conference & Exposition, Session 2268, 2001.
10. Zhao, J., “Teaching Finite Element Analysis as a Solution Method for Truss Problems in Statics”, Proceedings of the 2004 American Society for Engineering Education Annual Conference & Exposition, Session 1566, 2004.
11. Cook, Nathan H.. Mechanics and Materials for Design. New York: McGraw-Hill Book Company, 1984.