

A Finite Element Module for Undergraduates

Joseph J. Rencis, Hartley T. Grandin, Jr., William O. Jolley

University of Arkansas/Worcester Polytechnic Institute/The Gillette Company

Abstract

This paper presents a study module that is incorporated into a formal introductory undergraduate level course on finite element theory and practice. The module consists of an Integrative Project and Homework Exercises based upon sophomore level education in mechanics of materials. The objective of the module is to support the teaching of the finite element method and to emphasize assumptions and limitations in the application of the technique.

The Project centers on a simply supported beam with geometric discontinuities. This beam is investigated using a commercial finite element code in five different phases. Each phase uses a different solution model consisting of a hand calculation, beam, two-dimensional area, and three-dimensional solid elements. The solution from each phase is compared to the solution from traditional mechanics of materials beam theory in terms of the following: weight and center of gravity, deflection, and stress. Static failure theories, stress concentrations and a redesign are also considered. The approach of a comparative solution to a problem using different element types has not been considered in any finite element textbook to date and very few books consider stress concentrations and failure theories.

The Homework Exercises involve solving similar, yet smaller problems using commercial software and verifying the finite element solutions with the mechanics of materials solutions. The five homework exercises considered here include: a truss, an L-bracket with re-entrant corner, a plate with a notch, a thick-walled pressure vessel, and a pin joint connection.

The project and some of the homework exercises reinforce the following: an understanding of finite element theory, an understanding of mechanics of materials theory, a knowledge about the physical behavior and usage of each element type, the ability to select a suitable element for a given problem, and the ability to interpret and evaluate finite element solution quality. Emphasis is strongly placed on the importance of verification. The project and several of the homework exercises also illustrate common major conceptual mistakes made by students and, often, by practitioners using commercial software.

What is the Finite Element Method?

The finite element method (FEM) is a mathematical technique that simulates physical behavior by means of a numerical process based on piecewise polynomial interpolation applied to the controlling fundamental equation. The method has been used extensively during the past thirty years in industry and is now a standard engineering tool for both analysis and design.

Engineering analysis has always faced the challenge of modeling complex real problems by replacing the real problems with carefully designed, yet easily manipulated simpler problems which obey the same fundamental principles. Today, the finite element method is one of the most widely used methods of formulating simplified or idealized problems in the fields of solid and fluid mechanics, and heat transfer. Years of experience with the method have shown that by understanding the fundamentals of the technique, real complex systems can be modeled with a high degree of reliability.

It is important to emphasize, however, that the reliability of the process is highly dependent on the skill of the engineer in the application of the method. Modern finite element developments have become very sophisticated, and the available software developed for the user has become very easy to use. It has become more important than ever to insure that the analyst, in his/her search for the best modeling method, correctly uses the tools available.

What Type of Education is Required to Carry Out a Proper Element Analysis?

When FEM first appeared in the 1960's it was introduced into the engineering curriculum at the graduate level. As the method and computer technology matured, FEM was introduced at the undergraduate level in engineering and engineering technology programs, even in some two-year technology programs. Graphical user-friendly interfaces (GUI) have significantly reduced the complexities of the actual application of FEM software such that engineers with education equal to or less than the bachelors degree are using the technique today. In contrast, ten years ago, specialists did a majority of FEM analyses, mostly educated at the masters or doctoral level [1] due to the method's technical complexity and to the command line pre-processing requirements.

Finite element courses in academia at the undergraduate and graduate levels in engineering programs are mainly theoretical in nature. Although some students and practitioners have taken an FEM course at the undergraduate and/or graduate level, many individuals have only been introduced to FEM in a two to five day *training* course. These training courses enable an individual to 'build a model' and have the program run successfully to yield some output. However, these software-training courses fall short of teaching the supporting theory. This has led to misuse of finite element technology where, "Today, new users tend to believe that any results that look good are probably right [2]." Therefore, a person eager to use newly acquired software skills and lacking a good grasp of fundamental FEM theory and mechanics of materials is the most dangerous user! It is paramount that students and practicing engineers learn to be critical of their

results and not get into the bad habit of accepting computer-generated answers on faith. Therefore, it is essential for students and practitioners to be *well-educated*, not well-trained, in applied fundamental finite element theory and mechanics of materials.

The authors feel that to carry out a proper finite element analysis one *must* have a grasp of the problem area (mechanics of materials in this paper) and an understanding of classical analysis tools! Articles that say, “You need not know everything about FEA to successfully run today's analysis software [3],” are typically written by vendors who are trying to only sell their software. The author's feel that this is true to some point, but training is not sufficient. *Education is!* The finite element method is closely related to theory because the method is largely a way of implementing theory. Therefore, it is important that the user specifically understand the widely used (and widely misused) methods and formulas of mechanics of materials. The assumptions and restrictions underlining analysis tools that are incorporated in finite elements must also be understood! For students and practitioners, the main reason to study mechanics of materials and finite elements is that assumptions and restrictions are revealed enabling individuals to decide when and *when not* to use a particular theory or procedure. Only then can one correctly address questions that must be answered. What physical actions are important? Is the problem time independent? Are there nonlinearities? What are the boundary conditions? How will the results be checked?, etc. Finite element analysis is much more than geometric modeling, pre-processing, and running an analysis (button pushing)!

This paper presents a method to educate an individual in the fundamentals of finite element theory and practice using sophomore level mechanics of materials through one project and several homework exercises. First the undergraduate course at WPI is discussed and then proposed educational learning objectives are presented. Next the integrative project is discussed in-depth, and then five homework exercises are considered.

Undergraduate Finite Element Course at WPI

Finite elements (ME4512 Introduction to Finite Element Method) at the Worcester Polytechnic Institute (WPI) is a seven-week junior/senior level course that meets four class sessions per week for a total of twenty-eight class sessions (fifty minutes per session). The course is taught by the second author at WPI and focuses on stress analysis applications due to the short time duration of the course. The textbook by Logan [4] is used and the topics covered include the following element types: spring, truss, beam, plane stress/strain, axisymmetric, three-dimensional and plates. The course grade is based on two exams (50%), homework (25%), and two projects (25%) involving design insights and application of a commercial FEM code. The projects a majority of the time, are done in a group of two yet have been done individually. The reader should note that the project and homework exercises can be done using any commercial code. Two, labs are scheduled outside of class time and the first lab focuses on the truss and beam elements and the second on two-dimensional elements. The students are required to learn the other element types on their own through the homework exercises and projects. The

course typically enrolls twenty students and consists of approximately ninety-percent mechanical engineering majors and the rest come from civil engineering. One project and a few homework exercises considered in this course are discussed in this paper. In the next section the educational objectives of this course are discussed.

Educational Objectives of the Finite Element Course

This paper is in response to the national awareness that most engineers are well trained, but *not* particularly well educated in the fundamentals of the finite element method. With the significant number of publications available [5] on the theory, development and usage of the finite element method, it may be rather difficult for an instructor to identify an effective plan of study. The educational objectives for a course depend on whether the student or practitioner is a user or a researcher/developer of the technology:

- *User.* The user needs to learn the proper use of the finite element method for the solution of complex problems. This will require fundamental understanding of theory.
- *Research/Developer.* The researcher/developer needs a thorough understanding of the finite element method theory in order to develop new and, perhaps, extend the existing methodologies and/or develop or modify a finite element software code.

As educators in today's technological society we must recognize that over 99% of the students and practitioners will fall in the user category due to accessibility of the technology in the university and industry environments.

The educational objectives proposed in this paper will focus more on the user but will emphasize significant understanding of finite element theory and mechanics of materials theory. Without a solid background in *fundamental finite element theory* and *mechanics of materials theory*, a user of commercial packages is left guessing and hoping for the best. A thorough understanding of the theory would surely benefit the user. However, it is left primarily to the researcher/developer. The *eight proposed educational objectives* for a FEM course that focuses on stress analysis applications are as follows:

1. *Mechanics of Materials Theory.* Understand the fundamentals of mechanics of materials theory.
2. *FEM Theory.* Understand the fundamental basis of the finite element theory.
3. *FEM Element Characteristics.* Know the physical behavior and usage of each element type commonly used in practice.
4. *FEM Modeling Practice.* Be able to select a suitable finite element model for a given engineering problem.
5. *FEM Solution Interpretation and Verification.* Be able to interpret and evaluate finite element solution quality, including the importance of verification.
6. *FEM Assumptions and Limitations.* Be aware of the assumptions and limitations of FEM.

7. *FEM in Design.* Understand how FEM is used and applied in the design process.
8. *FEM Hand Solutions.* Be able to solve simple finite element problems by hand and compare the solution to that obtained by traditional mechanics of materials methods. (This objective will not be addressed in this paper.)

Using a variety of mechanics of materials problems in projects and homework exercises emphasizes basic use and misuse of commercial finite element software and satisfies these objectives. Verification of computer-generated solutions is paramount to the approach taken herein, and an elementary understanding of the theory behind the technique is strongly emphasized.

Project Definition and Goal

The undergraduate course requires students to carry out two projects. The second project is the focus of this paper and requires an investigation of the weight, center of gravity, displacements, and stresses in the simply supported beam shown in Figure 1. The time duration of this project is two-weeks. The project consists of five phases and the educational objectives each phase addresses are shown in Table 1. Phases 2 through 5 require the use of a commercial finite element code and each phase consists of a series of steps (questions). The steps from each phase are essentially the same and require a comparison of each element type to the mechanics of materials solution. Some of the questions illustrate common conceptual mistakes made by students and often by practitioners. The students are then required to write a professional engineering report on their findings.

The *goal* of this study module is to support the teaching of finite element theory and to emphasize assumptions and limitations in the application of the method to a simply supported beam, easily analyzed with traditional mechanics of materials techniques. The exercises in the study module illustrate to the student the importance of having a solid education in mechanics of materials theory as well as an understanding of the finite element method theory in order to produce and/or recognize valid engineering analysis using *any* commercial finite element code. The overall design concept of the study module is focused on the integration of fundamental mechanics of materials and practical finite element analysis.

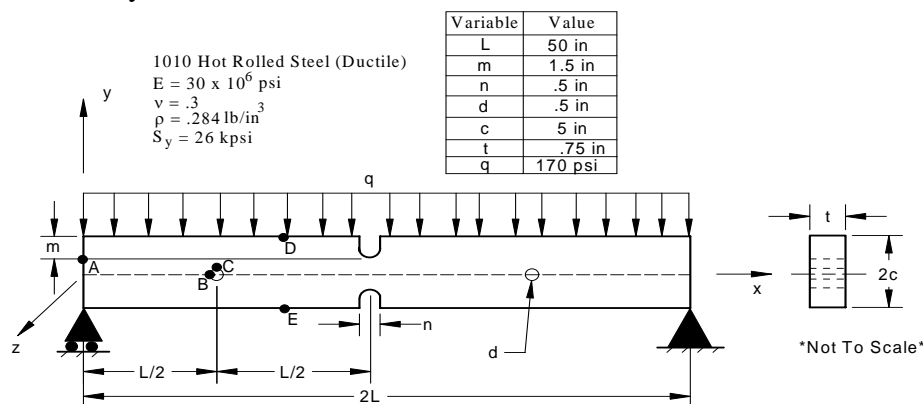


Figure 1. Simply supported beam with geometric and material data.

Table 1. Project phases and the educational objectives addressed by each.

Phase	Educational Objective(s) Addressed
1. Mechanics of Materials Solution*	1 & 5
2. Beam Element Models	2-6
3. Two-dimensional Element Models	2-6
4. Three-dimensional Element Models	2-6
5. Redesign	7

*Phase 1 is used to satisfy educational objectives 2 and 5 in phases 2-4.

Project Phases and Their Learning Objectives

There are five phases in this project as shown in Table 1 and each phase consists of a number of steps for the students to complete. The five phases are discussed in-depth as follows:

1. ***Mechanics of Materials Solution:*** Carry out a hand stress analysis of the simply supported beam using mechanics of materials principles. Students are required to first complete a hand stress analysis of the simply supported beam using sophomore level statics and mechanics of materials. The eight steps required to complete phase 1 are shown in Table 2. The educational objectives satisfied by this phase are 1 and 5 as shown in Table 1. The second author has found that this phase can take the students a significant amount of time since a review of mechanics of materials is needed.

Table 2. Eight steps of Phase 1: Mechanics of Materials Solution.

Step	Question
1a	Calculate the vertical deflection at the mid-span neglecting and including shear deformation. Ignore the notches and holes in your calculation. Is this problem considered to be a long (shallow) or short (deep) beam? Explain.
1b	Plot the shear and moment diagrams. Label the locations of maximum shear and moment.
1c	Calculate and plot the bending stress distribution throughout the depth of the section at $x=0$, $L/2$ and L .
1d	Calculate the maximum bending stress at the notch and hole using stress concentration factors.
1e	Calculate and plot the flexural shear stress distribution throughout the depth of the section at $x=0$, $L/2$ and L .
1f	Draw the actual and beam theory two-dimensional states of stress at points A, B, C, D and E labeled in Figure 1.
1g	Since the beam material is ductile, determine the von-Mises stress at points A, B, C, D and E. Determine the factor of safety and comment on its magnitude.
1h	Determine the total weight of the beam and its center of gravity.

This phase was selected as the first for two reasons. First, the students have been exposed to traditional mechanics of materials methods, and progress can be made immediately. Secondly, students will be generating results that should be directly comparable to those eventually obtained with the finite element analysis. The hope here is to try to instill the idea that an independent approximate analysis is vital in the critique of the validity of computer generated results. In the phases to follow, various finite element solutions are developed and each solution is compared to this first phase containing the mechanics of materials solution. The importance of having alternative methods for verification of finite element results is strongly emphasized through the comparison. Additional hand versus computer verification calculations are performed for the weight and center of gravity of the beam. If for some reason there is an obvious discrepancy between the hand calculation and the commercial code output, then they will know there is a problem requiring further checking (either with the hand calculation or the computer model). The students are continuously reminded that, “There are no wrong answers in FEA only wrong questions. If [students] can learn how to pose the questions correctly, they will get good answers [4].”

Figure 2 provides a portion of the required calculations for determining the lateral deflection at the midspan including shear deformation and the bending stress at the notch. The bottom notch is the critical location on the beam having the highest tensile state of stress. In the example, the calculated nominal stress at the notch is slightly above yield strength of the material. This has been done intentionally in preparation for a redesign of the beam in phase 5 as shown in Table 1. The finite element solution of this problem in phases 3 and 4 will show the very limited region of the stress intensity above the yield strength, and will show the student the rationale for not applying the stress concentration factor to ductile materials subject to static loading (i.e.: $K_t=1.0$). The reduction of the stress level through yielding of the material within the small region has no effect on the part. Only if subjected to load reversals will there be need to account for the stress raiser, i.e., fatigue.

Additionally, in step (1f) the students are required to determine actual and beam theory states of stress at various locations on the beam. This is where they will find the flexural shear stress formula says that there is a shear stress at point C in Figure 1, which is inconsistent with the actual shear stress being zero due to the free surface.

Vertical Deflection at Midspan Including Shear Deformation:

$$\delta_y = \frac{5qL^4}{16Etc^3} + \frac{3qL^2(1+\nu)}{5Etc}$$

$$\delta_y = \frac{5(128 \frac{lb}{in})(50in)^4}{16(30 \times 10^6 \frac{lb}{in^2})(.75in)(5in)^3} + \frac{3(128 \frac{lb}{in})(50in)^2(1+.3)}{5(30 \times 10^6 \frac{lb}{in^2})(.75in)(5in)}$$

$$\delta_y = .08893in \downarrow$$

Bending Stress at Notch:

$$\sigma_x = K_t \frac{M_z c}{I_z}$$

$$\sigma_x = (1) \frac{(160000lb - in)(3.5in)}{21.44in^4}$$

$$\sigma_x = 26122 \text{ psi Tension}$$

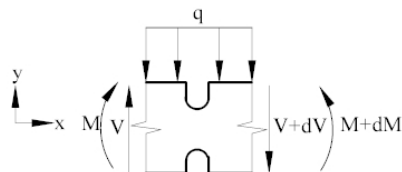


Figure 2. Phase 1 calculations of the maximum vertical deflection and bending stress.

2. **Beam Element Models:** Carry out a half-symmetry finite element analysis of the left-half (roller supported end) of the simply supported beam using three-dimensional beam elements. The first model will neglect shear deformation and the second model will include shear deformation. Students are now asked to complete the first finite element analysis of this project using a commercial code. The eight steps of phase 2 are shown in Table 3. The educational objectives satisfied by this phase are 2 through 6 as stated in Table 1.

Table 3. Eight steps of Phase 2: Beam Element Models.

Step	Question
2a	Construct three meshes of one, two, four and eight equal-size elements and neglect the hole and notch. State the number of nodes, total number of degrees of freedom, number of constrained degrees of freedom, number of unconstrained degrees of freedom, and the number of elements for each mesh.
2b	In tabular form compare the hand and FEM solution for the vertical deflection at the mid-span with and without shear deformation for each mesh. Check for convergence; plot the deflection versus the number of degrees of freedom in each mesh. What is the relative percentage error? Are the deflections and rotations considered to be small or large? Do these results violate elementary beam theory? Explain.
2c	In tabular form compare the hand and FEM solutions for the shear force at the nodes to 1b.
2d	Compare the hand and FEM solutions for the total weight and center of gravity of the structure in tabular form. What is the relative percentage error of the results?
2e	Consider that the left support is removed. Are the results obtained by FEM reasonable? Explain.
2f	Consider the eight-element model in 2a. At the left support now use a pinned support and reanalyze. Compare the vertical deflection at the mid-span to the vertical deflection obtained at the mid-span in 2b. Explain why you get the same solution.
2g	Do the beam elements reflect any sort of stress concentration effect at the supports? Explain.
2h	Do you expect the relative percentage error for displacements and the relative percentage error for stresses to be the same value, the same magnitude, or different? Why or why not?

The three-dimensional beam element was selected for three reasons. First, to illustrate that three-dimensional beam elements can be used to model one-dimensional space problems. Second, to demonstrate the need to be mindful of the third direction (not explicit), especially when the loading is entirely two-dimensional. A common error is the omission of the appropriate boundary conditions to suppress rigid-body motion of the structure. Third, to involve the student with all of the three-dimensional beam element geometric parameters: e.g., moments of inertia, torsional constant, shear areas (or shear deflection constant), etc., which most students and practitioners do not understand. Note in Figure 1 that the thickness (t) is much smaller than the depth of the beam ($2c$). Thus, if the moments of inertia are not defined correctly, the vertical displacement and stress will have a very different and incorrect value. The large difference in dimension was used intentionally to generate a large, but obvious error.

Although the beam element is formulated directly as a structural element, the force displacement analysis may or may not yield results identical to those of the mechanics of materials solution depending on the treatment of the distributed loading. If the distributed

loading is transformed into work equivalent nodal loads, the nodal displacement results will agree with the mechanics of materials solution. If the distributed loading is converted into statically equivalent lumped concentrated nodal forces, then the results will be approximate, the error being dependent on the number of equally sized beam elements. Since this exercise involves a distributed loading, the student will investigate which approach in the treatment of the distributed loading is coded into the commercial code and test for the solution convergence dependence on the number of elements, or number of degrees of freedom, in the model. This evidence is demonstrated in Figure 3.

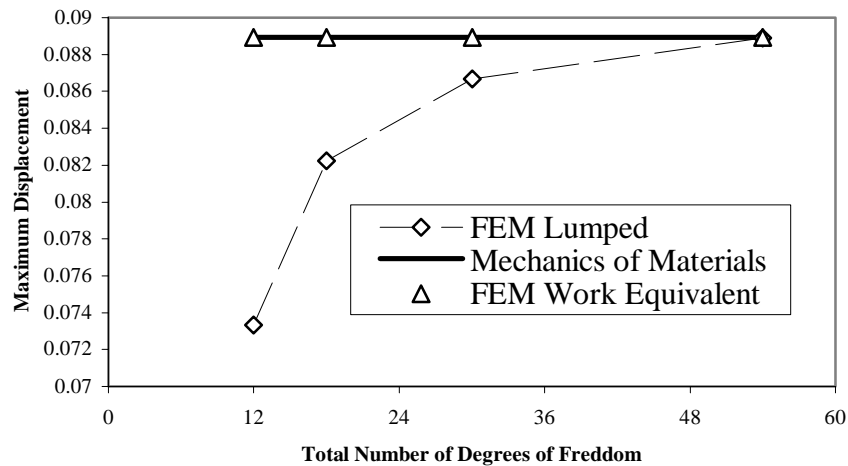


Figure 3. Convergence study for beam elements with shear deformation.

Taking advantage of symmetry the students are required to model only the left half of the beam. Students calculate the displacements and the bending moments at the same locations as required in the hand calculation model and are asked to compare their finite element results to their hand calculation results. Step (2a) requires that the students develop three finite element models for the analysis, i.e., a coarse, medium and fine model, to emphasize the importance of convergence. Continuous emphasis throughout the class is placed on the fact that the answer will only converge to the computer's representation of the problem, even if the model is not valid! Students are reminded that in step (2b) a comparison is done by neglecting and including shear deformation in the FEM models and comparing to the respective hand solutions. Many students and practitioners do not know what it means to model the problem in accordance to long beam and short beam theory. They commonly apply long beam theory with no regard given its limitations.

Figure 4 provides an example of the results for the mid-span displacement in the beam and the mid-span bending moment using the half-symmetry beam element model with shear deformation. The convergence of displacement and bending moment has been examined; the convergence of flexure stress is ignored because it is directly proportional to the nodal bending moment. Due to the fact that the beam element model software calculates the stresses from the nodal internal bending moments and shear forces using the traditional mechanics of materials relationships and since it does not account for the reduced moments of inertia and stress concentrations at the notches and holes, the students must perform these calculations. Step (2d) requires a calculation of the beams

weight and center of gravity. The students are reminded that this is one way to check if a model was properly defined. Furthermore, this check is very important when considering gravitational loading and when carrying out vibrational and dynamic analyses.

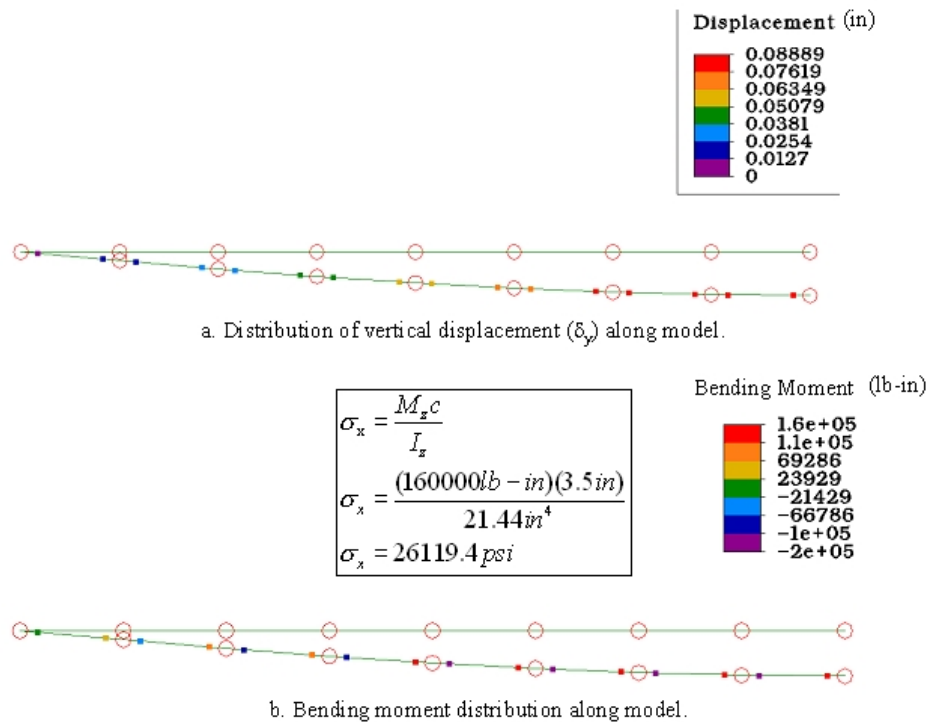


Figure 4. Results of the beam element analysis.

The final three steps, (2e) through (2g), bring in conceptual mistakes commonly made by students, and practitioners using commercial software. Step (2e) asks to remove the left hand support (roller) and rerun the analysis. From this exercise students learn to be cautious engineers, for some commercial finite element codes will solve this problem without yielding any error or warning messages indicating that the structure is unstable (presence of singularity). In fact the computer may actually yield a solution! In this case the displacement solution will be extremely large. However, the displaced screen plot is qualitatively correct. A cautious engineer will always be suspicious of a solution and carefully examine it. Step (2f) requires a change in the left support of Figure 1 from a roller to a pin demonstrating that there is no change due to beam theory assumptions. Finally, step (2g) asks if there is any type of stress concentration effect present at the support in the beam element.

3. **Two-dimensional Element Models:** Carry out a half-symmetry finite element analysis of the left-half (roller supported end) of the simply supported beam using four-noded quadrilateral elements. The thirteen steps of phase 3 are shown in Table 5. The educational objectives satisfied by this phase are 2 through 6 as stated in Table 1. The steps are very similar to Phase 2 for the beam element. After constructing and analyzing their two-dimensional finite element model of the simply supported beam, the students are again asked to compare the results to the solutions obtained in Phases 1 and 2. Only important concepts will now be discussed.

Table 5. Thirteen steps of Phase 3: Two-dimensional Element Models.

Step	Question
3a	Determine if the problem is plane stress or plane strain. Explain why you selected this problem type.
3b	Construct the following three meshes using four-noded quadrilaterals elements: coarse mesh, medium mesh and fine mesh. For each mesh state the element type used, number of elements, number of nodes, total number of degrees of freedom, number of constrained degrees of freedom and number of unconstrained degrees of freedom. Make sure that you have selected a fine mesh that has converged for stress.
3c	Compare the hand and FEM solutions from 3b, 1a, and 2b for the vertical deflection at the mid-span of the beam (mid-depth) for each mesh in tabular form. Check for convergence; plot the deflection and bending stress versus the number of degrees of freedom. What is the relative percentage error? Does your finite element analysis account for deflection due to shear deformation? If so, is shear deflection significant? Why?
3d	Compare the hand and FEM solutions from 3b for the maximum bending stress at the locations stated in 1c in tabular form. What is the relative percentage error? Does the stress compare well at every point along the beam? Why?
3e	Compare the hand and FEM solutions for the maximum bending stress at the notch and hole in 1d and 2d in tabular form. What is the relative percentage error? Also compare the factor of safety. Comment on the magnitude.
3f	Compare the hand and FEM solutions for the maximum shear stress at the locations stated in 1e and 2e in tabular form. What is the relative percentage error? Does your finite element analysis account for deflection due to shear deformation? If so, is shear deflection significant? Why?
3g	Compare the two-dimensional state of stress at points A through E in 1f to the state of stress at the same points using the fine finite element mesh in 3b. Draw the state of stress for the two-dimensional FEM results. Does this state of stress at each point correspond to the actual or beam theory in 1f?
3h	Using the fine mesh in 3b, compare the yield strength S_y to the maximum von Mises stress. Has the beam yielded? What is the factor of safety? Comment on the magnitude of the factor of safety.
3i	Compare the hand and FEM solutions for the total weight and center of gravity of the structure in <i>tabular form</i> . What is the relative percentage error of the results?
3j	Consider the finely meshed model in 3b. At the left support now use a pinned support and reanalyze. Compare the vertical displacement at the mid-span to the vertical displacement obtained at the mid-span in 3c. Explain why you get a different solution. How does this solution compare to 1a?
3k	Consider that the uniformly distributed load q is doubled in magnitude. How much will the mid-span vertical deflection increase or decrease? Explain. Do not carry out a finite element analysis for this step.
3l	Do the two-dimensional elements reflect any sort of stress concentration effect at the supports? Explain. How does this compare to the beam element model in Phase 2?
3m	Do you expect the relative percentage error for displacements and the relative percentage error for stresses to be the same value, the same magnitude, or different? Why or why not?

Step (3b) focuses on constructing finite element models of various mesh sizes from coarse, medium, to fine, thereby making sure that students know how to determine a suitable mesh for a given problem. From the comparison of solutions students find that they cannot trust the results of just one analysis. They must construct multiple models refining the mesh until their changes in desired quantities are within a specified or

reasonable tolerance. They find that when displacements are of primary concern, a relatively coarse mesh is suitable for accurate results. However, when stresses are of primary concern a much finer mesh is required to investigate the distribution of stress in the beam, primarily in the area of geometric discontinuities. This fundamental and universal fact of finite element analysis is revealed in a practical approach through the solution of the project problem. In step (3b) the students discover that deformations due to shearing effects are considered in the two-dimensional formulation. They learn that deformation due to shear is important, yet typically neglected because generally the governing stresses in long and slender beams are bending stress and as a result shear deformation is relatively insignificant when compared to bending deformation in long beams. In steps (3d) through (3h) the important lesson is that the two-dimensional model determines the actual stress distribution throughout the entire beam; therefore, stress concentration factors do not need to be applied. Also in these steps the students are required to obtain a plot of the von Mises stress distribution from the converged model, and obtain a factor of safety for the beam. Students discover that the beam is poorly designed for the required loading and geometry because the factor of safety is slightly less than one therefore a redesign is required. Figure 5 provides the results for the maximum displacement and bending stress at the notch using a half-symmetry model.

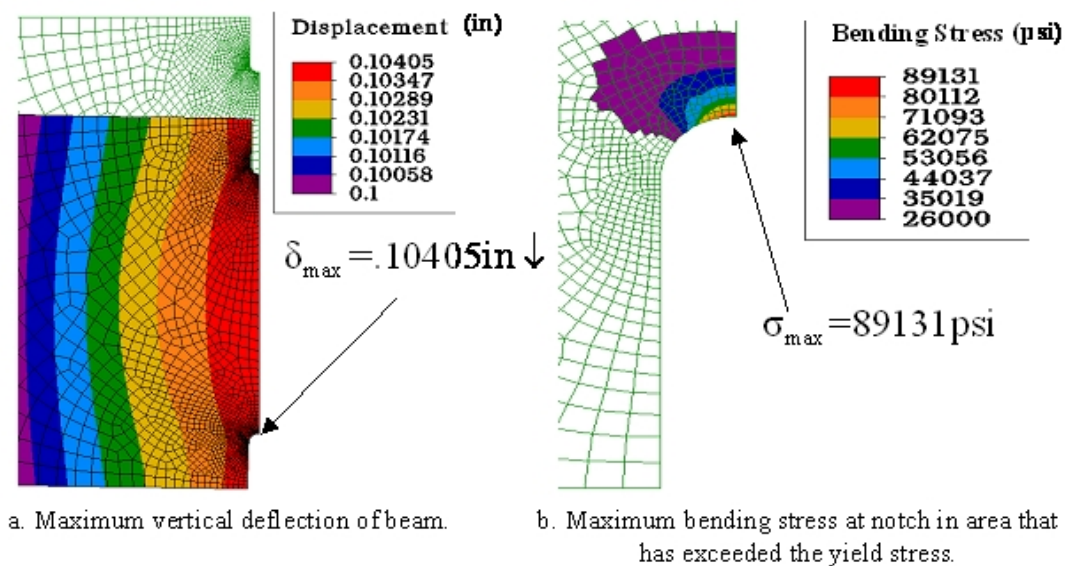


Figure 5. Phase 3 results of the two-dimensional analysis.

The students have discovered from the two-dimensional and also from the three-dimensional analyses that the notch at the bottom of the beam gives rise to an increase in the flexure stress considerably above the calculated nominal. The ratio of the maximum stress at the notch to the calculated nominal stress is defined to be the stress concentration factor, K_t . From the two-dimensional analysis, this stress concentration factor was determined to be equal to 3.4. A comparison with the published value of 3.2 indicates a very close agreement and further illustrates the reliability and the value of the finite element analysis.

Examination of the high stress at the notch reveals that the elevated stress above the nominal is confined to a very small region as shown in Figure 5. At this point, the

students are reminded of the following concepts from their design course background. If the beam loading is static, and if the beam material is ductile, the small region of stress above the yield strength will result in a one time very localized yielding and a slight redistribution of the stress level. The yielding will not result in a noticeable deformation. Thus, for static loading, the calculated nominal stress is a satisfactory measure of the loading intensity, and the beam in this exercise would have a factor of safety of unity. However, if the beam loading fluctuates with time, and/or if the beam material is brittle, the stress concentration factor must be applied to the calculated nominal stress as dictated by a fatigue failure theory.

Steps (3j) through (3k) again focus on conceptual mistakes commonly made by students, and practitioners using commercial software. Step (3j) requires the left support to be changed from a pin to a roller. Since the support is located on the bottom of the beam, the vertical displacement (and horizontal) will change since the bottom fiber of the beam is in tension, thereby applying horizontal forces that affect the response throughout the beam. This is clearly a misrepresentation of the support conditions yielding a pin-pin supported beam instead of a pin-roller supported beam. However, if the supports were placed on the centroidal longitudinal axis of the beam, then the results would be consistent to the displacements calculated in Phase (1a). Step (3k) signifies that by doubling the distributed load (q) in Figure 1, the vertical displacements will double. Finally, step (3l) shows that the two-dimensional model does account for stress concentration effects at the supports. Students learn that without its correct detailed boundary conditions the solutions in the vicinity of the supports are unreasonable and cannot be taken as accurate results in accordance with Saint-Venant's Principle from elementary mechanics of materials.

4. **Three-dimensional Element Models:** Carry out a quarter-symmetry finite element analysis of the left-half (roller supported end) of the simply supported beam using eight-noded brick elements. The nine steps of Phase 4 are shown in Table 6. The educational objectives satisfied by this phase are 2 through 6 as shown in Table 1. Due to the length of this project and the complexity of developing three-dimensional models, the length of each question was reduced in this phase.

Table 6. Nine steps of Phase 4: Three-dimensional Element Models.

Step	Question
4a	What are the symmetry boundary conditions? Describe and provide a screen capture of the model with applied boundary conditions clearly identified.
4b	Construct a coarse and a fine mesh using the eight-noded brick elements. For each mesh state the element type used, number of elements, number of nodes and the total number of degrees of freedom. Make sure that you have selected meshes that converged on stress.
4c	Compare the vertical displacement at the midspan of the three-dimensional models to that of the hand solution, beam element, and two-dimensional element models in tabular form. What is the relative percentage error?
4d	Using the fine mesh for the beam element, two-dimensional element, and three-dimensional element model, compare the solutions for the maximum bending stress at the notch and hole in tabular form to the hand solution in 1d. What is the relative percentage error?

4e	Compare the hand and three-dimensional FEM solutions for the total weight and center of gravity in tabular form. What is the relative percentage error of the results?
4f	Compare the two-dimensional state of stress at points A through E in 1f to the state of stress at the same points using the fine finite element mesh in 4b at the symmetry plane. Draw the state of stress for the three-dimensional FEM results.
4g	Carry out a finite element analysis using four-noded tetrahedrals instead of the eight-noded quadrilaterals. How does the von-Mises stress at the notch compare to the eight-noded hexahedral element model?
4h	Do you expect the relative percentage error for displacements and the relative percentage error for stresses to be the same value, the same magnitude, or different? Why or why not?
4i	Of all the analyses you have carried, i.e., mechanics of materials, beam element model, two-dimensional model and three-dimensional model, which one did you prefer? Why? Which one is practically the most efficient from a model creation and analysis point of view?

This phase of the project introduces the students to three-dimensional finite element analysis. Why look at a three-dimensional model? The main reason is to show that for two-dimensional problems, it is an excessive use of computational resources. Due to the rapid increase in computing technology, mainly CAD systems, many students and practitioners naively construct three-dimensional solid finite element models all the time, instead of using a simpler model when appropriate. This is well summarized in *Building Better Products with Finite Element Analysis*, stating that, “The technology has become so accessible that it is actually “hidden inside” CAD packages [2].” Furthermore, Kurowski [7] states that “The technology gets the nod even when hand calculations or physical testing would be faster, less expensive, and more accurate than FEA.” The past statement does not just apply to three-dimensional models, but using finite elements as the preferred analysis tool.

In this phase the students are supplied a solid model (solid iges file) and are required to go through the process of turning that solid model into a finite element model of eight-noded brick elements. Questions (4a) through (4f) in Table 6 are similar to Phases 2 and 3. The students quickly discover that three-dimensional analysis requires much more time to create the model and to obtain a converged solution compared to the beam and two-dimensional models. The analysis results reveals that there is no variation of displacement or stress throughout the beam thickness. From this exercise students learn that a three-dimensional analysis is not required since the computational cost of analyzing the beam is not justified when a simpler model is sufficient. Figure 6 includes a portion of the results from the quarter-symmetry three-dimensional beam model. The results depict the bending stress at the bottom notch. In fact, the mesh shown is not fine enough to capture the stress in the vicinity of the notch, yet has many more degrees of freedom compared to the two-dimensional model. The results of all four different analyses are summarized in Table 7. One can now compare the different strategies used to solve the problem and draw conclusions on which type of analysis would have been more efficient from a model creation and analysis standpoint.

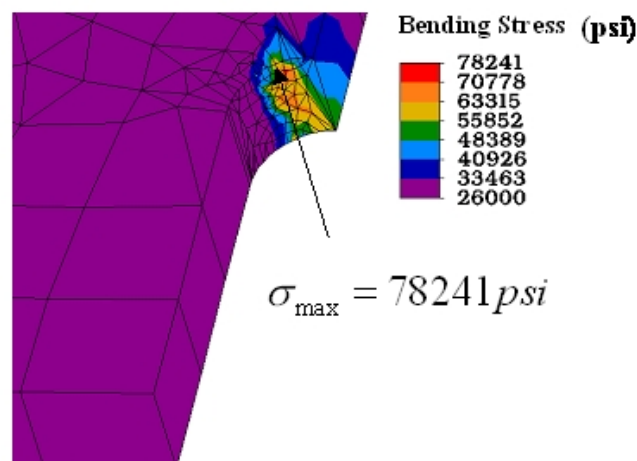


Figure 6. Phase 5 results showing the yield strength has been exceeded at the notch.

Table 7. Comparing the different analysis types for the simply supported beam.

Phase	Maximum Displacement at Midspan (in)	Maximum Bending Stress at Notch (psi)
1. Mechanics of Materials	.08893	26119
2. Beam Element Models (w/Shear, and Work Equivalent Loading)	.08893	26119
3. 2-D Element Models	.10405	89131
4. 3-D Element Models	.10225	78241
5. Redesign	.00755	8620

Step (4g) requires an analysis using a four-noded tetrahedral model and then comparing the von Mises stress at the notch to the eight-noded brick element model. In this exercise students find out why the eight-noded brick element is preferred over four-noded tetrahedral element in practice due to its inability to determine accurate stresses. Step (4i) requires the students to select which analysis is preferred, i.e., mechanics of material, beam element model, two-dimensional model or three-dimensional model. This step also requires them to decide which analysis was the most efficient from a model creation and analysis point of view. A hand solution would be the preferred analysis technique and from a finite element point of view the beam element model is sufficient.

- 5. Redesign:** *Redesign the simply supported beam based on a desired factor of safety and deflection limit.* This phase requires just a redesign of the beam and therefore, no questions are posed like in the previous phases. This phase satisfies educational objective 7 as stated in Table 1. A redesign component was considered herein since in modern mechanical design it is rare to find a project than does not require some type of finite element analysis. For example, to be more competitive, companies have also moved finite element analysis (FEA) from the later stages of the design cycle into the early design stage [2,7]. In this phase FEM is used to demonstrate how it can improve a design.

The students have discovered from the previous phases that the simply supported beam has failed due to strength and therefore if selected and used as a final design failure of the beam would be inevitable. This design criterion is that the beam has a factor of safety of three and the vertical deflection is limited to $L/180$, where L is the half-length of the beam in this case. Furthermore, the following items are held fixed in the design: material, beam length, notch geometry and hole geometry. Using fundamental concepts of design and static failure analysis the students redesign the beam satisfying these design specifications. Students discover that when trying to satisfy deflection requirements, the strength requirements are generally satisfied as well. Students find that the redesigned beam is much deeper (98%) than the initially proposed design and learn that shearing deformation is now an important quantity to investigate, and may not be negligible at all. Figure 7 shows a plot of the von Mises stress distribution around the bottom notch. The initial design failed having a factor of safety of 0.3 and the redesigned beam has a factor of safety of 3.1. The students are also required to provide a sketch of the beam with the redesigned dimensions. It should be noted that the design was based on the maximum von Mises stress at the notch. However as discussed in Phase 2 the design could be based on the stress away from the notch.

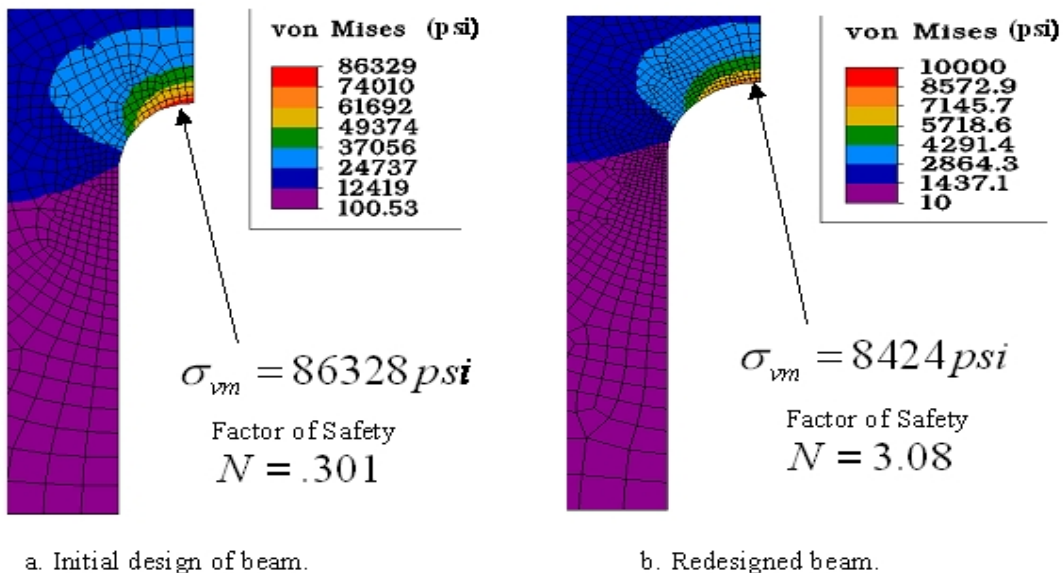


Figure 7. Beam redesign.

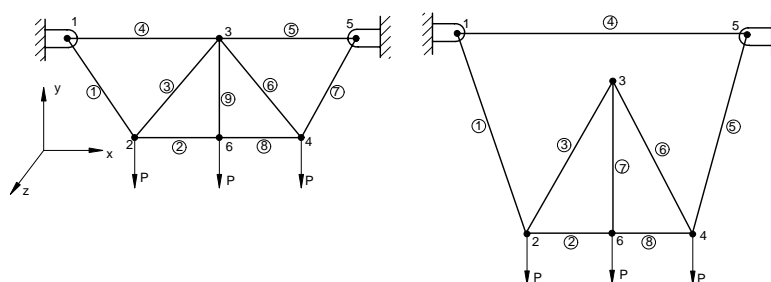
Five Homework Exercises

The homework exercises typically involve solving smaller problems using a commercial software and then verification of the FEM solution using mechanics of materials. The homework exercises reinforce the eight educational objectives previously discussed. The homework also illustrates common conceptual mistakes made by students and practitioners. Many of the homework assignments are rather untraditional from the perspective of FEM textbooks. Almost all FEM textbooks have homework exercises that require the theoretical development of an element or the solution of a simple problem by hand. In this section the five homework exercises discussed are shown in Table 8 along with their corresponding educational objectives.

Table 8. Five Homework exercises and the educational objective addressed by each.

Homework Exercise	Educational Objective(s) Addressed
1. Truss	1-6
2. L-Bracket	1,4,5
3. Plate with Notch	4
4. Thick-walled Pressure Vessel	1,3-5
5. Pin Joint Connection	1,3-5

The first example is a two-dimensional truss shown in Figure 8a that is modeled using three-dimensional truss elements. This problem can be used to ask many interesting questions that contain common mistakes made by students and practitioners alike. First, the three-dimensional nature of the truss element causes major problems since many individuals forget to fix the translational degree of freedom in the z -direction at all nodes to suppress the three global rigid body modes, i.e., translations in z -direction, rotation about x -axis and rotation about the y -axis. The authors have even found that commercial software tutorials also neglect the z -direction! Second, removing the support at node 5 will result in a structure that is not tied down from global rigid body rotation about the z -axis (as discussed in Phase 2 of the project). Third, removal of element ③ will result in a quadrilateral shape that causes a singularity due to the local rigid body mode, i.e., a triangular shape is a stable truss configuration but a quadrilateral shape is not. Fourth, removal of vertical element ⑨ with node 6 still present in elements ② and ⑧ yields a singularity since there is no resistance in y -direction (zero stiffness) perpendicular to elements ② and ⑧. Fifth, one major concept that is discussed considerably in the FEM course is convergence, i.e., refining the mesh. However, in this case introducing nodes in between a member introduces an artificial pin that is not present in a real truss member. This again leads to a singularity. Sixth, a common mistake that students make when creating a truss is to define a truss element that is not connected at the nodes, e.g., there is a node associated with elements ③ and ⑥ at node 3, but there is no node in element ④ as shown in Figure 8b. This leads to a discontinuity (gap) in the truss where node 3 separates from element ④.



a. Problem definition. b. Exaggerated deflected shape showing a gap in truss.

Figure 8. Three-dimensional truss homework exercise.

The second homework exercise is the L-bracket shown in Figure 9. The exercise is to find the maximum von Mises stress in the L-bracket and to determine if the part fails when subjected to the uniformly distributed load of 1500 N/mm. A commercial finite element code is used to carry out a convergence study by solving the problem using

successive mesh refinements as shown in Figure 10 (smaller elements are used in the vicinity of the re-entrant corner). A plot of the von Mises stress versus the number of degrees of freedom is shown in Figure 11. From this graph the students find that the stress will never converge. The reason is that theory of elasticity states that an infinite stress arises at a re-entrant corner. This is a common mistake that the authors have found being done by practitioners where they chase a stress that can never be obtained. Even more common is that students and practitioners do not carry out a proper convergence study and simply use the value of the von Mises stress for a given mesh. This type of application reinforces how important it is for students and practitioners to have an understanding of finite element theory and mechanics of materials theory. Furthermore, educating students on these slight yet often overlooked problems in finite element analysis instills a strong sense of practical and fundamental modeling skills. If the bracket had a fillet at the corner of interest then the stress will converge. This exercise demonstrates that “A lack of understanding finite element fundamentals can introduce the potential for erroneous stresses and deflections even in simple classical examples [8].”

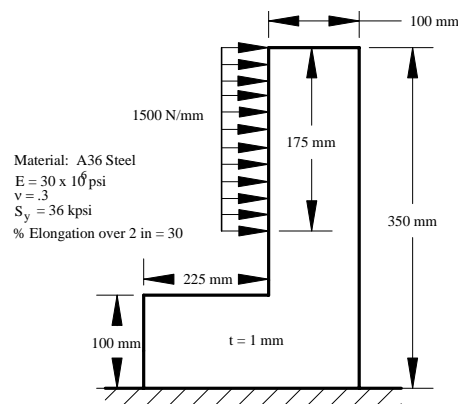


Figure 9. L-bracket homework exercise problem definition.

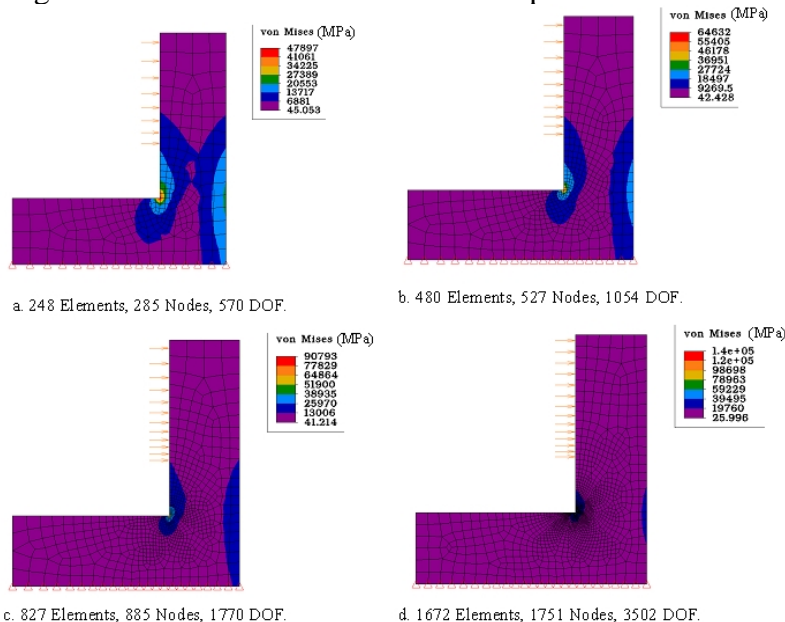


Figure 10. L-bracket finite element plane stress solution for the von Mises stress at the re-entrant corner for four 4-node quadrilateral meshes.

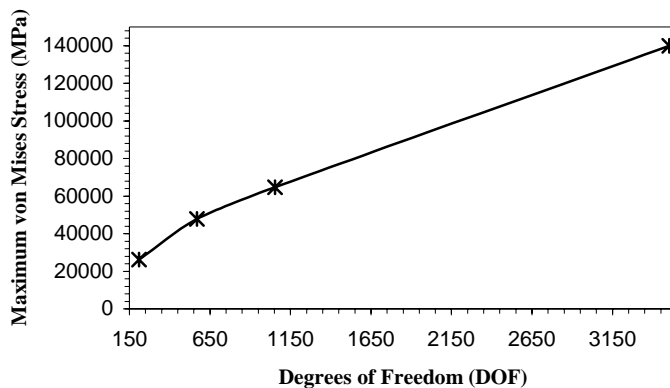


Figure 11. L-bracket non-convergence study of von Mises stress at re-entrant corner.

The third homework exercise improves the students' grasp of defining the correct boundary conditions, which tend to be one of the most misunderstood concepts of developing a finite element model. A plate with a notch is shown in Figure 12a and is subjected to purely force boundary conditions. The problem is modeled using two-dimensional elements. The important point is that the problem shows no type of supports (constraints). Therefore, the requirement is to define the appropriate boundary conditions that will tie down the plate from rigid body motion, i.e., translation in x , translation in y and rotation about the z . As can be seen in Figures 12b and 12c there are two choices. The first option shown in Figure 12b shows the supports placed to the left-hand side of the plate so that the plate can bend and therefore will not affect the stress concentration at the notch. The distance from the left end to the notch was selected as the depth of the plate so the supports will not affect the stress concentration at the notch in accordance to Saint-Venant's principle. The second option shown in Figure 12c accounted for the symmetry in the problem. The pin was placed at the bottom since this is the furthest location away from the stress concentration at the notch along the line of symmetry.

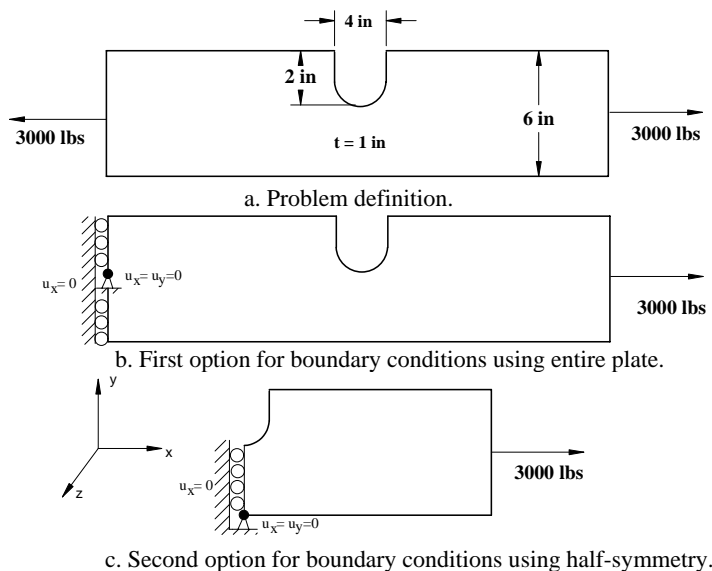


Figure 12. Plate with a notch homework exercise.

The fourth homework exercise is a thick-walled pressure vessel subjected to an external pressure of 5000 psi. The students are asked to find the solution for radial displacement, and hoop and radial stresses in the vessel four different ways using mechanics of materials, two-dimensional model, axisymmetric model and a three-dimensional model. The von Mises stress distribution and factors of safety are also calculated in this homework exercise. Finally, a comparison of the four different solution methods are required and shown in Figure 13. This homework exercise places strong emphasis on verifying computer-generated solutions.

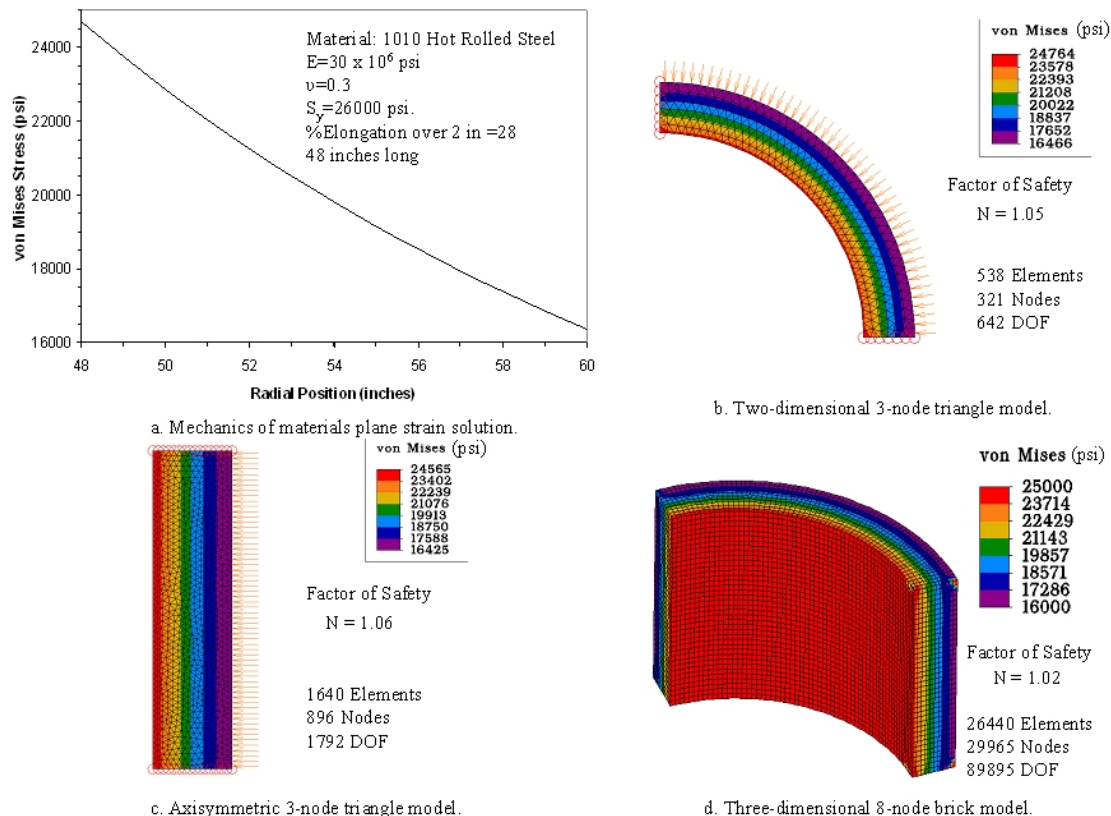
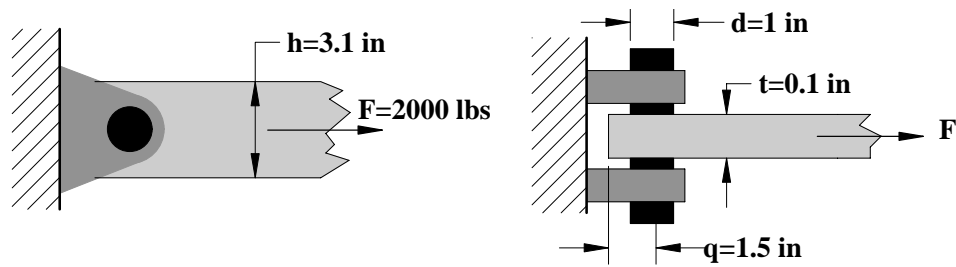


Figure 13. Thick-walled pressure vessel homework exercise using three different element types and mechanics of materials.

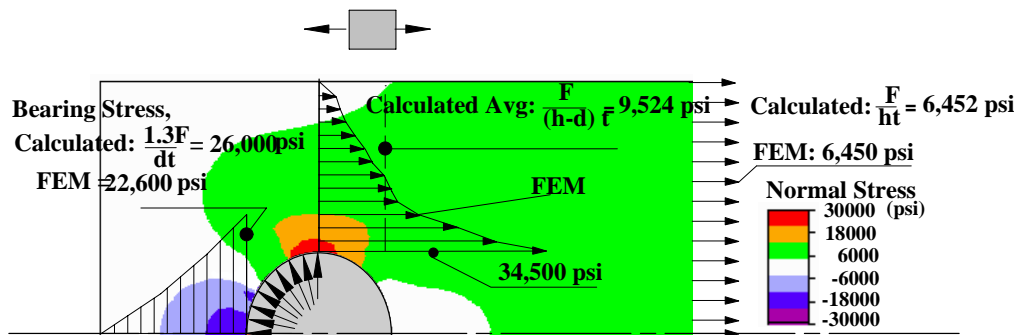
The fifth homework exercise is commonly found in the introductory chapter on stress in a mechanics of materials textbook. Consider the double shear pinned connection shown in Figure 14. In an introductory chapter on stress you will be asked to calculate the normal stress in the member, the bearing stress of the pin acting on the plate and the tearout shear stress in the plate. Figure 15 shows a comparison between the results obtained from a two-dimensional FEM stress analysis assuming plane stress and using four-noded quadrilateral elements. The pin surface was modeled as a fixed boundary and gap elements were placed between the pinned fixed boundary and nodes on the plate. This example shows how mechanics of materials solutions compare quite well to the finite element solution based on theory of elasticity.



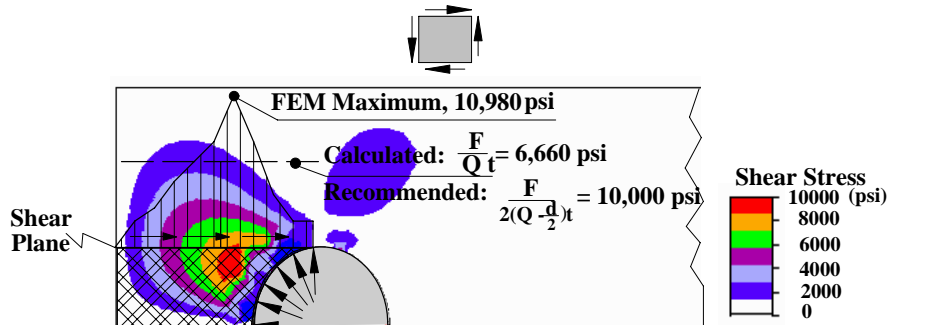
a. Top view.

b. Side view.

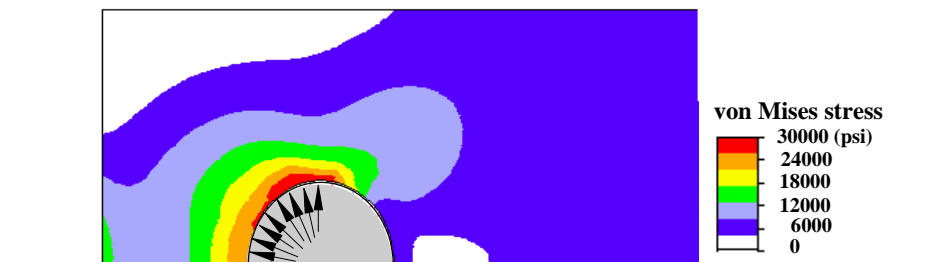
Figure 14. Double shear pinned connection homework exercise.



a. Cross section normal stress.



b. Tearout shear stress.



c. von Mises stress.

Figure 15. Half-symmetry model of plate with hole comparing mechanics of materials and finite element solutions.

Student Feedback and Course Modifications

Student feedback has been very positive regarding the learning experience of the project and homework exercises. A problem when the project was first introduced into the course was that the students were confronted by many challenges all at once. Students would get so overwhelmed on one aspect of the project, e.g., shear deformation that they never adequately answered other challenges, e.g., plane stress versus plane strain or geometric discontinuities. These problems were reduced/eliminated by modifying the classroom topics to address the projects and homework exercises. Furthermore, in the project assignment guidance is provided on where to look in their statics and mechanics of materials textbooks. The topics are not conventionally found in a traditional FEM course or textbooks. One topic includes applying stress concentration factors to truss and beam elements, but not to two-dimensional, axisymmetric and three-dimensional elements. Other topics include static failure, fatigue failure, shear deformation in beam elements, and element rigid body modes. The homework exercises can also be carefully assigned so a topic can be addressed on a small scale. Comments from the students have been very positive since the topics address the practical usage of FEM. One issue that we still have not been able to solve is that students will wait until last minute to start the project; however, a 25% grade reduction per day has minimized this problem.

Conclusions

This paper has presented a study module that includes a project and homework exercises that can be incorporated into an introductory undergraduate course on finite element theory and practice, however, it can easily be used in an introductory graduate-level course. The major reason why fundamental finite element theory and mechanics of materials theory are strongly emphasized is to avoid the danger of using finite elements as a black box. The project and homework exercises reinforce an understanding of mechanics of materials theory, finite element theory, knowledge about the physical behavior and usage of each element type, the ability to select a suitable element for a given problem, and the ability to interpret and evaluate finite element solution quality. Verification is an important component of the module. The module illustrated common major conceptual mistakes made by students and, often, by practitioners using commercial software. The discovery of these common pitfalls by the student results in a much more memorable experience versus a classroom lecture. The authors feel that this module will provide the student a first step in obtaining the proper *education*, not training, to use the best modeling method to solve real complex systems with a high degree of reliability.

References

1. Thilmany, J., "Analyzing Up Front," *Mechanical Engineering*, Vol. 122, No. 10, October 2000.
2. Adams, V. and Askenazi, A., *Building Better Products with Finite Element Analysis*, OnWord Press, Santa Fe, NM, 1999.
3. Gonzalez, U. "Smarter FEA Software Unburdens Users," P.J. Dvorak (ed.), *Machine Design*, pp. 120-126, March 1, 2001, <http://www.machinedesign.com/>.

4. Logan, D.L., *A First Course in the Finite Element Method Using Algor™*, Second Edition, Brooks/Cole, Pacific Grove, CA, 2001.
5. "Information Retrieval on Finite Element Books," *Department of Mechanical Engineering, Linköping Institute of Technology, Sweden*, <http://ohio.ikp.liu.se/fe/>.
6. Kurowski, P., "3.5 When Good Engineers Deliver Bad FEA," in *Basics of Design Engineering: CAD/CAM, Machine Design*, <http://www.machinedesign.com/>.
7. Roth, G., "The Value of Early Analysis (Part 1)," *BENCHmark*, National Agency for Finite Element Methods and Standards (NAFEMS), pp. 15-18, July 2000, <http://www.nafems.org/publications/>.
8. Kurowski, P., "Easily Made Errors Mar FEA Results," *Machine Design*, September 13, 2001.

JOSEPH J. RENCIS

Joseph J. Rencis is currently Professor and Head of the Department of Mechanical Engineering at the University of Arkansas. From 1985 to 2004 he was in the Mechanical Engineering Department at the Worcester Polytechnic Institute. His research focuses on the development of boundary and finite element methods for analyzing solid, heat transfer and fluid mechanics problems. He serves on the editorial board of *Engineering Analysis with Boundary Elements* and is associate editor of the *International Series on Advances in Boundary Elements*. He is currently writing the fifth draft of an introductory mechanics of materials textbook with the first co-author. He has been the Chair of the ASEE Mechanics Division, received the 2002 ASEE New England Section Teacher of the Year and is a fellow of the ASME. In 2004 he received the ASEE New England Section Outstanding Leader Award and in 2006 the ASEE Mechanics Division James L. Meriam Service Award. He currently serves as the Vice Chair of the ASME Department Heads Committee. He received his B.S. from the Milwaukee School of Engineering in 1980, a M.S. from Northwestern University in 1982 and a Ph.D. from Case Western Reserve University in 1985. V-mail: 479-575-3153; E-mail: jjrencis@uark.edu.

HARTLEY T. GRANDIN, JR.

Mechanical Engineering Department at Worcester Polytechnic Institute. He has authored the textbook *Fundamentals of the Finite Element Method* that was published by Macmillan in 1986. Since his retirement from WPI in 1996, he teaches a mechanics of materials course each year and is currently writing the fifth draft of an introductory textbook with the author. In 1983 he received the WPI Board of Trustees' Award for Outstanding Teaching. He received his B.S. in 1955 and an M.S. in 1960 in Mechanical Engineering from Worcester Polytechnic Institute and a Ph.D. in Engineering Mechanics from the Department of Metallurgy, Mechanics and Materials Science at Michigan State University in 1972. E-mail: hgrandin@rcn.com and hgrandin@wpi.edu.

WILLIAM O. JOLLEY

William O. Jolley current works for The Gillette Company in Boston, MA. He is currently a graduate student in Mechanical Engineering at Worcester Polytechnic Institute. He received his B.S. in Mechanical Engineering in May of 2002. His interests in research have focused on using finite element methods for analyzing solid mechanics problems, vibrations, and accident reconstruction. Ultimately he is planning on pursuing a Ph.D. in Mechanical Engineering and a career in academia. E-mail: William.Jolley@gillette.com.