Valparaiso University
# ValpoScholar

Engineering Faculty Publications & Patents

College of Engineering

6-2016

# Self-Paced, Problem-Solving Approach to Teaching Finite Element Analysis in Strength of Materials

Anne Raich
*Valparaiso University*, anne.raich@valpo.edu

Follow this and additional works at: http://scholar.valpo.edu/engineering_fac_pub

Recommended Citation

Raich, A. ( June 2016). Self-paced, problem-solving approach to teaching finite element analysis in strength of materials. Proceedings from the ASEE Annual Conference and Exposition '16: Teaching & Learning Statics and Mechanics of Materials. New Orleans, LA.

# Self-paced, Problem-solving Approach to Teaching Finite Element Analysis in Strength of Materials

**Dr. Anne Raich, Lafayette College**

Prof. Raich is an Associate Professor of Civil and Environmental Engineering at Lafayette College. Her teaching interests are in structural mechanics and analysis, structural design, and computational methods and applications. Prof. Raich received her Ph.D. from the University of Illinois and previously worked as an assistant professor at Texas A&M University and as a structural engineering consultant.

# Self-Paced, Problem Solving Approach to Teaching Finite Element Analysis in Strength of Materials

## Introduction

The finite element (FE) method is used widely by both civil and mechanical design professionals. Therefore, providing opportunities for students to gain an understanding of FE theory and to gain practice modeling and analyzing engineering problems using professional FE analysis software is critical even for undergraduates. The content of a traditional Strength of Materials course was revised to provide students with a basic FE skill set, as well as to improve their understanding of how to use FE analysis as part of a design process. This change was motivated by the need to streamline the sequence of upper-level design courses, while still providing students with the opportunity to learn the FE modeling and analysis skills needed to be successful in subsequent courses and in design project work. The change also increased the exposure students have to computational engineering tools, which is a stated desire of ABET and our departmental external advisory boards.

This project studied the effectiveness of the approach taken to integrate FE modeling and analysis content into four sections of Strength of Materials. The sections had student enrollments of 20, 22, 30 and 18 and were taught by two different faculty members. The added content was constructed with the following objectives: a) improve the student's understanding of specific stress, strain and deformation topics covered in the course by integrating experiences using a FE analysis program, b) provide students with a basic understanding of FE theory, c) provide students with the skill set needed to model and analyze combined load problems using a FE analysis program; and d) provide students with an understanding of how element type, mesh size, support conditions, and other modeling decisions may impact FE analysis results.

Previous studies have sought to incorporate FE modeling and analysis content as early as the freshman year into the engineering curriculum. However, implementing these approaches in Strength of Materials courses often requires students to spend considerable time learning FE theory before being able to use commercial FEA programs. A few studies have stressed the benefit of students learning how to work directly with FEA programs to model and analyze familiar mechanical systems without needing to first drag them through the FE theory.[1,2] The primary focus of the study presented is to help students gain appreciable FE modeling and analysis skills. A secondary focus is placed on gaining a basic understanding of FE theory, especially related to potential modeling issues due to the use of different element types and mesh sizes and the presence of stress concentrations. The ability to successfully focus on modeling and analysis with novice students is strongly supported by the easy-to-use graphical user interfaces provided by the geometry and mechanical workspaces in ANSYS Workbench 15.0.[3]

In fall 2015, significant in-class and out-of-class course material was added to all four sections of the course. The added FE content consisted of eight hours of in-class recitation time and thirty-two hours of out-of-class self-paced instruction time. The out-of-class, self-paced instruction time was organized using stand-alone tutorials that presented specific topics in a visual format. Each recitation was run as a problem solving session in which students worked individually to complete ANSYS-based homework problems. In all the tutorials and homework, students were

required to compare hand calculations with ANSYS results in order to reinforce concepts covered in class lectures and to help verify model accuracy. In addition, students were required to complete a combined load design project, which involved designing an eccentrically-loaded bar system to meet specific stress limits using both ANSYS$^©$ Workbench$^{TM}$ 15.0[3] and separate hand calculations. It should be noted that all material covered in the tutorials and corresponding recitations was material added to the Strength of Materials course. No course material was removed or reduced in breadth or depth from the in-class course sessions in this implementation.

**Implementation**

Students receive one course credit, which is equivalent to four credit hours, for Strength of Materials in our curriculum. Students typically take this course in the fall semester of the sophomore year. In previous semesters, the course was taught with three in-class contact hours and an informal fourth hour that required up 14 hours of student work outside of class, often in the form of additional homework or projects. In fall 2015, the fourth hour was officially scheduled as a once-a-week, in-class recitation period. During the first six weeks of the semester, the recitation time was used to help students build problem solving skills by asking students to apply concepts learned in class to more complex problems. No homework was assigned during these first six recitation sessions.

For the remaining eight weeks of the semester, recitation sessions were sequenced to specifically help students learn how to use the finite element analysis (FEA) program ANSYS$^©$ Workbench$^{TM}$ 15.0[3]. The scope of material covered over these eight weeks was defined to ensure that students would be able to model and analyze elements subjected to axial, shear, bending, and combined loads. Each week the students were required to:
1) <u>Prepare</u> for each ANSYS recitation session by completing an assigned ANSYS tutorial <u>before</u> attending the corresponding ANSYS recitation session. No grade was assigned for completing the assigned ANSYS tutorials.
2) <u>Work</u> during the recitation session either on completing the assigned ANSYS tutorial due to errors or other issues occurring during their outside-of-class prep work or on completing the assigned ANSYS homework problems.
3) <u>Complete</u> the assigned ANSYS homework before the start of the next recitation session.

Each self-paced ANSYS tutorial focused on a specific topic. These topics are summarized in Table 1. The sequence of the tutorials was constructed to build the knowledge and skills required to successfully model and analyze an eccentrically-loaded bar system to meet specific stress limits in ANSYS Workbench 15.0[3]. This requirement was added by extending an existing semester Combined Stress Project to now include ANSYS modeling and analysis student work. The Combined Stress Project details were presented in Tutorial 6. Homework 6 was assigned after completing Tutorial 6, but the final project report was not due until the end of the semester. Therefore in the following discussion, information about Tutorials 1, 2, 3, 4, 5, and 7 are presented first. Then, information related to Tutorial 6 is covered in a separate section that provides details concerning the Combined Stress Project requirements.

Table 1. Schedule of FEA Topics Covered in Recitation Sessions

| Recitation | Tutorial | Topic |
|---|---|---|
| 1 | 1 | Intro to FEA Modeling – Axial Stress and Deformation |
| 2 | 2 | FEA Modeling & Analysis – Bending Stress and Deformation |
| 3 | 3 | FEA Theory – Bending/Truss Optimization |
| 4 | 4 | FEA Modeling & Analysis – Combined Load Analysis |
| 5 | 5 | FEA Modeling & Analysis – Focus on Principal Stress Analysis |
| 6 | 6 | FEA Modeling & Analysis – Intro to Combined Load Project |
| 7 | 7 | FEA Modeling & Analysis – Focus on Stress Concentrations |
| 8 | None | FEA Modeling & Analysis – Modeling Issues/Element & Meshes |

The specific sequence of the self-paced tutorials and assigned homework problems was designed to achieve several objectives as mapped out here: Objective a) improve the student's understanding of specific stress, strain and deformation topics covered in the course by integrating experiences using a FEA program – Recitation 1, 3, 4, and 5; Objective b) provide students with a basic understanding of FE theory – Recitation 2 and 8; Objective c) provide students with the skill set needed to model and analyze combined load problems using a FE analysis program – Recitation 4, 5 and 6; and Objective d) provide students with an understanding of how element type, mesh size, and support conditions may impact FEA results – Recitation 7 and 8

Implementing this schedule of the eight self-paced tutorials, in-class recitation sessions, and accompanying homework assignments added significantly to both faculty-student contact hours and to the student work expectations outside of class. To help collect feedback, students were asked to complete on-line surveys supported by a Qualtrics website after each ANSYS tutorial. The online survey provided a mechanism for students to report an estimate of the time they required to complete each tutorial. This collected data is presented in Table 2 and also graphed in Figure 1 to help identify the average time students required to complete each tutorial.

Table 2. Student Estimated Time Required to Complete Tutorials (% of responses)

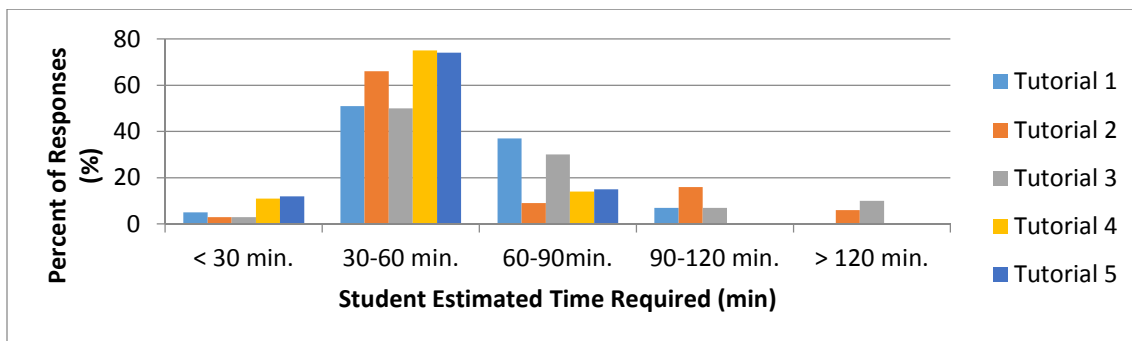| Tutorial | < 30 min. | 30-60 min. | 60-90 min. | 90-120 min. | > 120 min. |
|---|---|---|---|---|---|
| 1 | 5 | 51 | 37 | 7 | 0 |
| 2 | 3 | 66 | 9 | 16 | 6 |
| 3 | 3 | 50 | 30 | 7 | 10 |
| 4 | 11 | 75 | 14 | 0 | 0 |
| 5 | 12 | 74 | 15 | 0 | 0 |



Figure 1. Estimated Time Required to Complete Tutorial 1, 2, 3, 4 and 5

From the collected responses, it is clear that many students finished the tutorials within 60 minutes, and that almost all students finished the tutorials within 90 minutes. The online survey did not ask students to report their average hours worked on each homework assignment. However based on the complexity of problems solved, the reporting requirements for each assignment and discussions with students about time spent on outside-of-class work, the time required for each homework was proportional to the time required to complete each tutorial. Therefore, the student time-on-task required to support the integration of the ANSYS FEA content into the course during fall 2015 was estimated as 12 hours of out-of-class time to complete Tutorials 1, 2, 3, 4, 5, and 7; 8 hours of in-class recitation work; 12 hours of out-of-class time to complete Homework 1, 2, 3, 4, 5, and 7; and 8 hours of out-of-class time to complete the Combined Stress Project tasks and prepare the report. The total added student time-on-task was 8 hours of in-class time and 32 hours of out-of-class time. The average time per week for eight weeks of the semester was one hour in-class and four hours out-of-class.

**Overview of ANSYS Tutorial and Homework Assignments**

This section provides details about the ANSYS tutorials and homework assignments that were designed for the fall 2015 semester. These assignments define the skills students were expected to learn and also define the scaffolding of increasingly complex problems that were introduced to students in the assignments. The complete ANSYS Workbench 15.0[3] Tutorial and Homework Assignments from fall 2015 are provided at the following website: sites.lafayette.edu/raicha under the Strength of Materials course information.

*Tutorial 1 - Intro to FEA Modeling – Axial Stress and Deformation*
This tutorial was structured to guide students through the process of creating their first model in ANSYS Workbench 15.0[3] and then through analyzing their model to obtain axial stress and deformation results. Students learned how to define the model geometry and assign material properties. They also learned how to assign a mesh and run an analysis. The introductory model was a solid steel bar that was fixed at one end and had an axial concentrated load applied to the free end as shown in Figure 2.
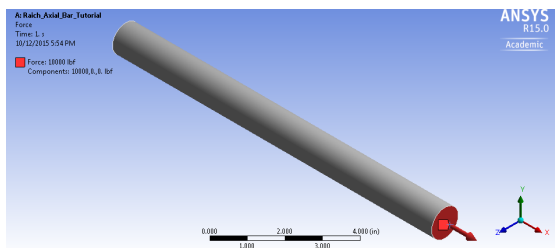


Figure 2. View of Axial Bar Model Created in Tutorial 1

Tutorial 1 also introduced students to several different ANSYS result viewers, including viewers for axial stress and deformation results. Before viewing any ANSYS-produced results, students were required to hand-calculate the bar axial stress and axial deformation. Students were then able to compare the ANSYS results obtained with their hand-calculated results to help confirm the accuracy of the model and their calculations. This comparison process also helped students

begin to identify regions that are subjected to local stress concentration effects. Other topics covered included viewing normal strain, lateral strain and Poisson's Effect.

*Homework 1 - Intro to FEA Modeling – Axial Stress and Deformation*
In the first homework assignment, students applied their new ANSYS skills to model and analyze two axial bars with different cross-sections: a rectangular, solid bar and a hollow tube. Each bar had an axial force applied to one end and a fixed support at the other end similar to the Tutorial 1 model. Views of these two models are shown in Figure 3.
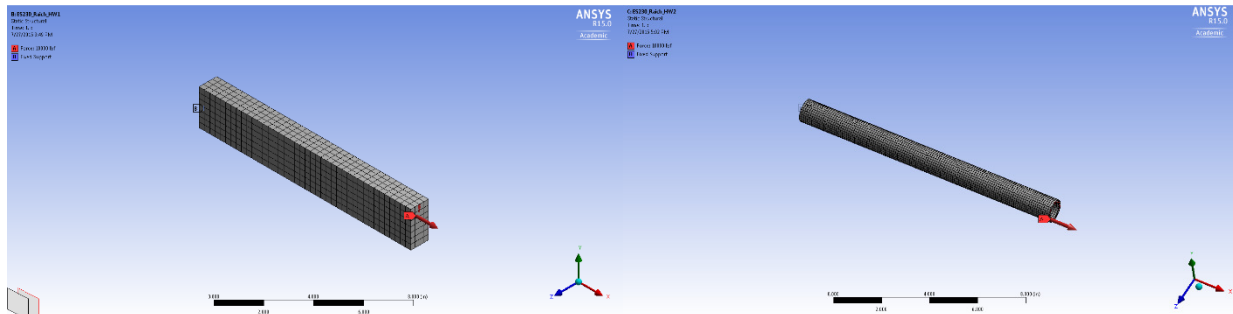


Figure 3. Views of Two Axial Bar Models Created in Homework 1

Following the process defined in the first tutorial, students first hand-calculated the normal stress, normal strain and total axial deformation in each axial bar using methods learned in-class. They then compared their hand-calculated result values to ANSYS provided results for stress, strain and deformation. Students were required to provide an explanation of whether they expected each result to be similar or not and to explain the reason for any difference in values.

*Tutorial 2 - FEA Modeling & Analysis – Bending Stress and Deformation*
In Tutorial 2, students learned how to draw a sketch of an I-beam shaped cross-section and how to assign that cross-section to beam elements to create a four-foot-long cantilever beam model that had a single concentrated load applied at its free end. These model features are shown in Figure 4. This tutorial also introduced students to two different beam bending theories that are used to calculated beam deformations. First the difference between 1D Beam Elements and 3D Solid Elements was discussed and then the behavior captured by 'Timoshenko Beam" elements and by 'Euler-Bernoulli Beam" elements was discussed. Although both beam theories accurately calculated bending stress, students learned through several examples that ANSYS will predict larger deformations than the deformations they calculated by hand for short beams.
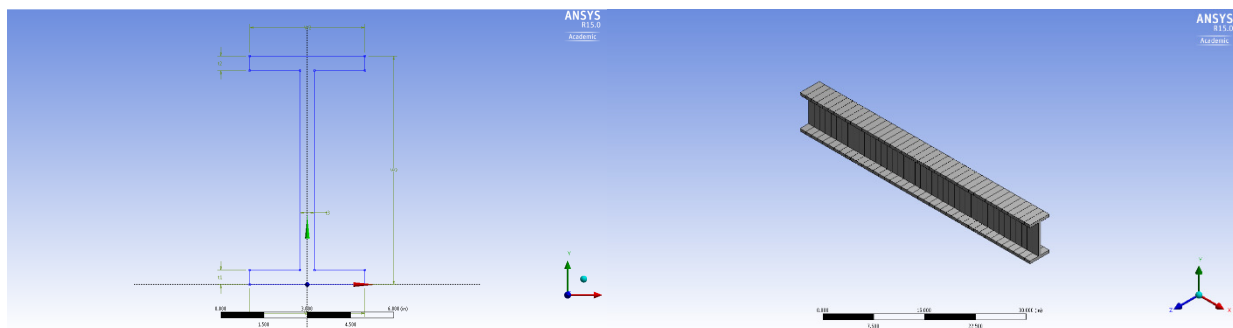


Figure 4. Sketch of Beam Cross-Section and View of Beam Model Created in Tutorial 2

After creating their model, students performed hand calculations to determine the maximum shear force and bending moment in the beam, the moment of inertia of the cross-section, the maximum bending stress in the beam and the maximum vertical deformation of the beam. Students then ran a comparison of the ANSYS results obtained with their hand calculated results to confirm the accuracy of their model and hand calculations. This comparison process reinforces the relative differences in stress values and deformation values that are expected for each beam case run using the two different beam theories.

To help assess the effectiveness of the tutorial in conveying beam theory information, students were asked several questions in an online survey conducted after Tutorial 2. Question 1 asked "The Timoshenko Beam Theory used by ANSYS considers the specific effect of which of the following when determining the beam's deflection. This specific effect is ignored in the Euler-Bernoulli Beam Theory we use for our hand calculations". A summary of the student responses provided to this question are provided in Table 3.

Table 3. Student Responses to Question 1 on the Online Survey for Tutorial 2 (% of responses)

| Bending Stress | 2 |
|---|---|
| Transverse Shear Strain | 28 |
| Axial Strain | 2 |
| Axial Stress | 0 |
| Moment of Inertia | 0 |

Question 2 asked "Since the Euler-Bernoulli Beam Theory that we use in our hand calculations ignore the effect above, the deformations you calculate by hand will be (Answer) deformation that ANSYS provides using Timoshenko Beam Theory". A summary of the student responses to this question are provided in Table 4.

Table 4. Student Responses to Question 2 on the Online Survey for Tutorial 2 (% of responses)

| Equal to the | 0 |
|---|---|
| Smaller than | 29 |
| Larger than | 3 |

Both sets of student responses to these questions indicate that the majority of students had acquired a basic knowledge of the difference between using these two beam theories in calculating beam deformation.

Student responses to a third online survey question, however, were extremely varied. Question 3 asked "In the tutorial you rotate the axis of bending of the initial beam cross-section by 90 degrees. If you did not rotate the beam element cross-section before running the analysis, the vertical deformation of this beam would be (Answer) and the maximum bending stress would be (Answer) than the values you obtained after rotating the cross-section by 90 degrees". The range of student responses included 28% responding larger and larger; 6% smaller and smaller; 25% larger and smaller; 16% smaller and larger, 19% smaller and the same; and 6% larger and the same. These responses were helpful in gaining an understanding that the students did not understand major and minor axis bending and also had difficult with two-part survey questions.

*Homework 2 - FEA Modeling & Analysis – Bending Stress and Deformation*
In this homework assignment students were required to apply their growing ANSYS skill set to model and analyze three different beam bending problems. Each problem was a modification of the beam model created in the Tutorial 2.  The first modification increased the beam length to create a slender beam.  The second modification used an aluminum material instead of steel and the third modification required the I-shaped, cross-section to bend about its minor axis instead of about its major axis.  Figure 5 provides views of the beam models for the first and third models.
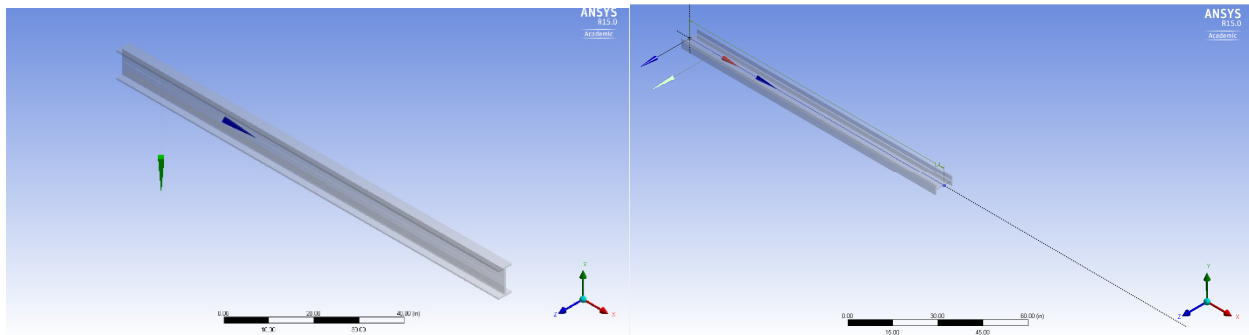


Figure 5. Views of Two Beam Models Created in Homework 2

Students first calculated the maximum shear force, maximum bending moment, maximum bending stress and vertical deformation for each of the three different beam configurations. Students then compared their hand-calculated results to ANSYS provided results for bending stress and vertical deformation.  Using the comparison information collected, students were required to discuss differences between stress and deformation values that occurred due to the beam configuration and beam theory used.  The use of ANSYS helped students gain a better understanding of overall beam behavior, how to locate regions of of maximum stress and defomation, and the effect of changing the beam length, material, and axis of bending on stress and deformation.

*Tutorial 3 - FEA Theory – Bending/Truss Optimization*
ANSYS has many tools that help designers determine near-optimal cross-sections, lengths, and loads for specific design problems.  In this tutorial students learned how to use the Direct Optimization Tool in ANSYS Workbench 15.0[3].  Although the beam model created in Homework 2 was used, the top and bottom flange widths of the cross-section were now considered as design variables.  In addition, the mass property and the maximum bending stress were set as optimization design parameters.  Search objectives were added to find the optimal flange width for the I-shaped cross-section that minimized the mass of the beam while meeting specific bending stress limits.  Students found a new design solution that had an optimal flange width and also confirmed that this new design produced the desired maximum bending stress results.  Based on survey responses collected after the completion of Tutorial 3, 97% and 93% of the students correctly stated that a larger flange width and a larger overall beam mass, respectively, were required if lower stress limits were imposed.

*Homework 3 - FEA Theory – Bending/Truss Optimization*
Students extended their design optimization skill set learned in Tutorial 3 to model and analyze a nine-bar truss model.  The students first created a square tube cross-section geometry and truss

layout and then analyzed the model to determine internal forces, internal stresses and truss deformations. Views of these model elements are shown in Figure 6. Students were then able to use the Direct Optimization Tool to determine a new truss depth that restricted the end vertical deformation of the truss to a specific value.
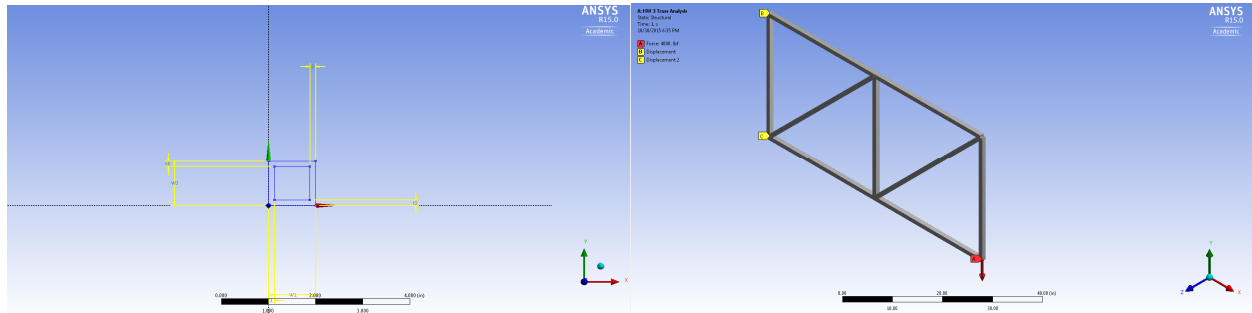


Figure 6. Sketch of Member Cross-Section and View of Truss Model Created in Homework 3

After analyzing the truss model, students performed calculations to determine internal forces and stresses in each truss member by hand and compared these results to the ANSYS results obtained to confirm that their model and calculations were accurate.

*Tutorial 4 - FEA Modeling & Analysis – Combined Load Analysis*
In this tutorial students modeled a bar that had an offset bend along its length, and therefore had both axial and bending stresses acting in the center portion of the bar. To develop this model students sketched the square cross section and then swept the cross-section along a specified path to create the required 3D bar profile. Views of these model elements are shown in Figure 7.
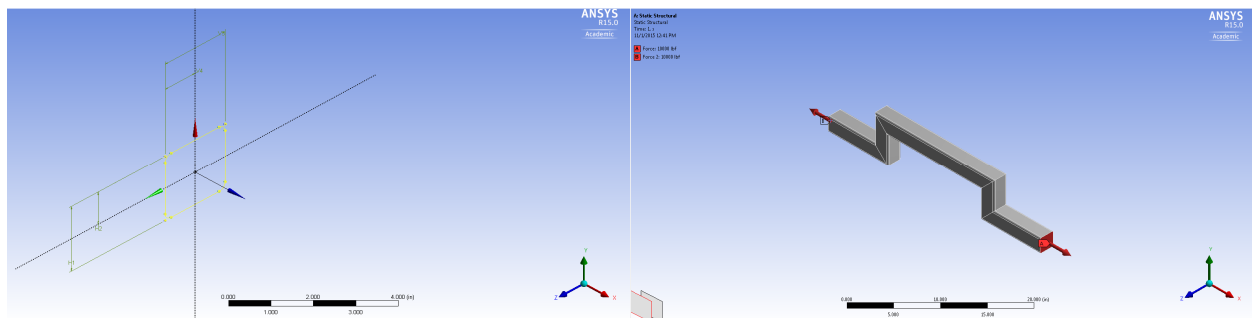


Figure 7. Sketch of Cross-Section and View of Eccentric Bar Model Created in Tutorial 4

Students also learned to track ANSYS results in order to collect information on total and directional deformations and normal stresses along the bar length for the whole model and at specific sections using defined section cuts. Although students could not calculate deformations by hand, they were required to compare hand-calculated normal stress values with ANSYS provided normal stress results at several section-cut locations.

In the online survey conducted after Tutorial 4 was completed, students were asked to self-assess their skill level in calculating the normal stress in a bar due to the combined loading by hand. The ratings reported by students were 50% stating "No Problem"; 50% stating "Could Do Hand Calcs With Assistance from Notes or Peers"; and 0% stating "Can't Do". To further differentiate

student knowledge at this point, students were asked whether the tensile normal stress was larger or smaller than the compressive bending stress in the bar modeled in the tutorial. Based on knowledge reinforced in the tutorial, 83% correctly answered this question.

*Homework 4 - FEA Modeling & Analysis – Combined Load Analysis*
Using the sketching and sweeping skills learned in Tutorial 4, students created a model of a compound bar having a solid, circular cross-section and subjected to a load that created bending moment, shear force, and toque at cross-sections near the fixed support. A view of the 3D model created is shown in Figure 8.
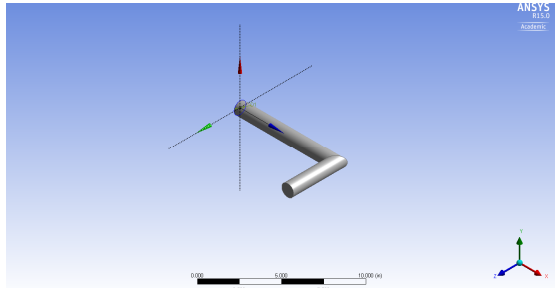


Figure 8. View of 3D Bar Model Created in Homework 4

After creating the 3D Bar Model students obtained ANSYS results for the directional bar deformation and also for the normal stress and shear stress occurring at specific sections due to the applied loading and bar geometry. Since the overconstrained fixed support conditions make a direct comparison of the stress levels difficult due to localized stress concentration and other effects, students learned how to take section cuts through the cross-section to obtain ANSYS results that could be compared to their hand-calculated results. Having a graphical user interface environment was ideal since students could visually reinforce their understanding of different stress distributions acting on different bar cross-sections.

*Tutorial 5 - FEA Modeling & Analysis – Focus on Principal Stress Analysis*
The focus of the fifth tutorial was to teach students how to interpret principal stress results and maximum shear stress results. The 3D-eccentric bar model created as part of Tutorial 4 was used for this purpose. Students were guided through how to view and interpret the ANSYS results for maximum and minimum principal stress and maximum shear stress. As in previous work, students were required to calculate specific stress values by hand. They were then able to view the results provided by ANSYS, which helped them visualize different states of stress acting on points on a cross-section. Students were able to practice finding principal stresses using Mohr's Circle for states of stress having only normal stresses acting at any point on the section plane cut. The students could easily compare the principal stress and maximum shear stress values obtained using Mohr's Circle with the principal stress and maximum shear stress values that ANSYS provides. This type of strain transformation process is an essential process in design.

The primary results engineers seek from a FEA program are often the maximum and minimum principal stress. Most failure theories are based on these stresses, so having the skill to obtain these results using ANSYS is very useful. To provide additional practice with principal stresses and stress transformation, students created and analyzed a 3D-bar bending model, which is

shown in Figure 9.  This 3D bar is subjected to loading that creates normal stress and shear stress at specific points on the bar cross-section.
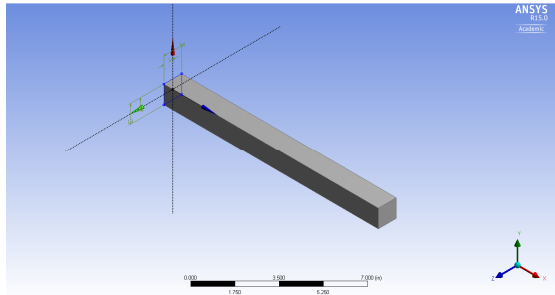

Figure 9. View of 3D Bar Model Created in Tutorial 5

*Homework 5 - FEA Modeling & Analysis – Focus on Principal Stress Analysis*
In Homework 5 students first calculated the principal stresses and maximum shear stresses using ANSYS and then compared those values with the ANSYS results obtained for the eccentric bar created in Homework 4 (Figure 8).  Using these results, students then calculated and compared the von Mises failure criteria stress values and determined if failure would occur at any of the points examined on a specific cross-section.  The process was partially self-guiding, since significant differences in stress values typically indicates possible errors in either the hand-calculations or the ANSYS model.

After completing Tutorial 5, students were asked the following questions, the answers to which had not yet been covered in class lectures or discussion.  Question 1: "For a 2D stress element with only a normal stress in the x-axis direction (i.e., sigma x = 3000 psi; sigma y = 0 psi; and tau xy = 0), the maximum shear stress found by stress transformation acts at an angle of 0 degrees, 45 degrees, or 90 degrees?" - 56% of students responded correctly.  Question 2: "For a 2D stress element with only a shear stress acting in pure shear (sigma x = 0 psi; sigma y = 0 psi; and tau xy = 250 psi), the maximum shear stress found by stress transformation acts at an angle of 0 degrees, 45 degrees, or 90 degrees?" – 53% of students responded correctly.  These results indicate that students gained some familiarity with transforming states of stress and about half of the students were able to successfully transfer that knowledge and apply it to other simple cases.

*Tutorial 7 - FEA Modeling & Analysis – Focus on Stress Concentrations*
The topics covered in this tutorial were selected to help students learn how to recognize and deal with a common modeling issues involving stress concentrations.  The initial model created was a flat plate with a circular hole as shown in Figure 10.  Students created a sketch of the cross-section with the hole and then extruded the cross-section to make the part.
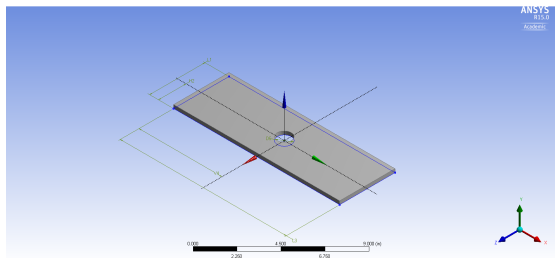

Figure 10. View of Flat Plate with Circular Hole Model Created in Tutorial 7

Students were tasked with learning how to compare the stresses ANSYS reported in regions located near the hole with the hand-calculated values they obtained using an approximate stress concentration factor. Students also compared the stress concentration results obtained for a plate made out of a non-yielding material and a plate made out of a yielding material. They learned through experience that modeling elastic-plastic material behavior helps to reduce errors in calculating stresses in areas that have stress concentrations in ANSYS.

At the end of Tutorial 7, students created a seat belt tab model that included elastic-plastic material behavior in ANSYS. A view of this model is shown in Figure 11. Using this model, students investigated the presence of stress concentrations at reentrant corners and in the region of the holes. Creating this model required sketching the tab geometry and extruding the part, in addition to assigning the material properties and loading.
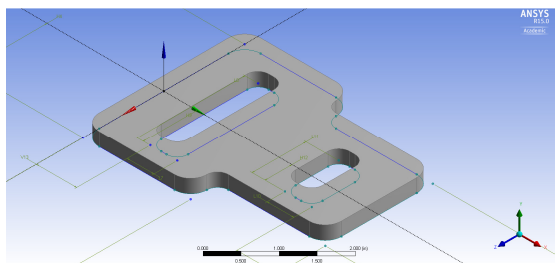


Figure 11. View of Seat Belt Tab Model Created in Tutorial 7

*Homework 7 - FEA Modeling & Analysis – Focus on Stress Concentrations*
This final homework assignment stressed the effect of design decisions on stress concentration levels in the seat belt tab model created at in Tutorial 7. For this process students investigated possible ways to modify the tab geometry so that stress concentrations were minimized. The three different configurations analyzed by students are shown in Figure 12. Students compared differences in directional deformation, overall volume of material and maximum von Mises Stress between the models. Students also were able to design a new seat belt tab geometry that used less material and that reduced the maximum von Mises stress by 50% or more.
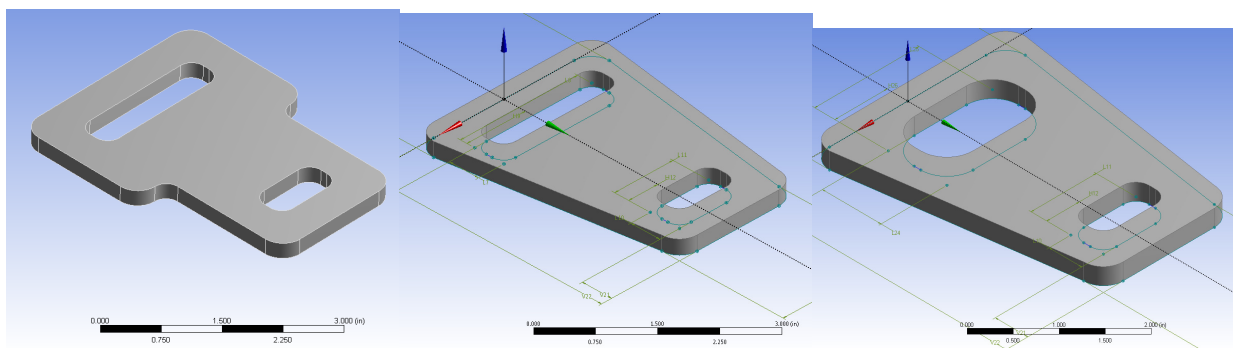


Figure 12. Views of the Three Seat Belt Tab Models Created in Tutorial 7

## Tutorial 6 - Combined Stress Project

The system analyzed consists of a sign of weight, W, supported by a sign post constructed using a hollow circular pipe cross-section. A resultant wind load force, P, acts on the sign as shown in

Figure 13. There were two main parts assigned in the project. Part B was added to the project for the first time in Fall 2015. Part A required the development of an Excel spreadsheet to determine the minimum acceptable wall thickness for the pipe cross-section if the pipe is made from a material with a specific yield stress and factor of safety. The calculations were based on a known pipe outside diameter, height and extension of the post; loading applied, material yield stress, and factor of safety. The design must consider the maximum combined effects of bending stress and shear stress acting on a section plane cut through the cross-section at points H and K. The failure criteria specified is the maximum distortion energy theory.
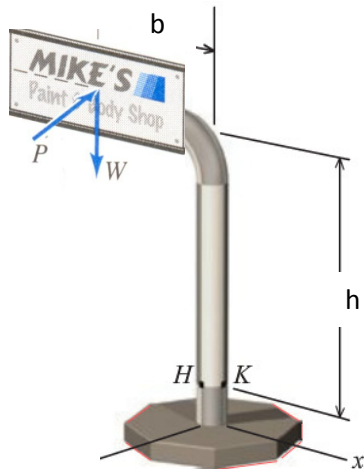


Figure 13. View of the Sign Post Geometry and Load/Support Conditions

Part B of the project required the development of an ANSYS model of the sign post. The model used the material defined, a hollow-tube cross-section having the minimum acceptable wall thickness determined using Excel, and the pole height and width dimensions defined. The primary ANSYS results reported were the Equivalent Stresses at the section plane cut through the cross-section at points H and K. The ANSYS model was graded on whether the cross-section geometry provided matched the minimum acceptable wall thickness; on the material properties defined; on the units used; on the correct load and support assignments; and on the equivalent stress results obtained using the appropriate result viewers.

The combined stress project was worth 5% of the final grade in Fall 2015. This grade was assigned in two parts: 85% of the grade was based on the accuracy of the Excel solution and the ANSYS model and analysis, while 15% was based on the report clarity. The specific grade items tracked in the assessment were the correctness of the Excel Spreadsheet, which reflects student understanding of hand calculations, and the correctness of the ANSYS model, which reflects student understanding of FEA modeling and analysis using ANSYS Workbench 15.0[3].

**Results**

Assessment was conducted in two of the four class sections run in fall 2015 in order to determine if adding significant FE content to Strength of Materials was effective in improving student

understanding of specific mechanics topics and providing a sufficient understanding of FE theory. Survey feedback was requested on a voluntary basis from 48 students. The surveys were administered online after each tutorial was completed. In the surveys, students self-assessed their confidence about recently acquired ANSYS skills. Grades given for the individual homework assignments were used to assess whether students could apply ANSYS skills to new problems. Also, final student grades on the semester design project provided a means to assess the competence of students in performing hand calculations and ANSYS analysis. Preliminary results indicate that students acquired a sufficient understanding of basic ANSYS modeling skills, but not of basic FE theory.

Students were asked to complete online surveys supported by a Qualtrics website. No grade was assigned for completion of the on-line surveys. Therefore, the completion rate depended on voluntary submittal of each survey by the students. Several reminders were needed to obtain the response rates shown in Table 5 for each Tutorial, since in some cases students began the survey but did not complete it.

Table 5. Student On-line Survey Response Rate

| Tutorial | Responses Started | Responses Completed | Possible Responses | Percent Response Rate |
|---|---|---|---|---|
| 1 | 46 | 41 | 48 | 85.4 |
| 2 | 39 | 32 | 48 | 66.7 |
| 3 | 35 | 30 | 48 | 62.5 |
| 4 | 38 | 37 | 48 | 77.1 |
| 5 | 44 | 34 | 48 | 70.8 |

After reviewing the data collected from the online surveys for the five tutorials, several general trends were found. Overall students in their self-assessments rated their skill level in modeling tasks higher than their skill level in viewing and understanding the analysis results provided by ANSYS. Table 6 provides an example of the response ratings for Tutorial 1, in which 29% to 46% of students rated their skill level in the "No Problem" category for the modeling-related skills. In comparison, Table 7 provides an example of the response ratings for Tutorial 1 in which 22% to 29% of students rated their skill level in viewing and understanding the analysis results related skills in the "No Problem" category. A mean response value is also provided in each table that is defined using 1 – No Problem, 2 – Could Do Again with Assistance from Tutorial, 3 – Would have Difficulty, and 4- Can't Do. For Tutorial 1, the average of the mean response rating for Model-Related skills was 1.76 while the average of the mean response rating for Results-Related skills was 1.82.
.
Table 6. Student Self-Assessment of Modeling Skills after Tutorial 1

| Skill | No Problem | Could Do w/ Tutorial Assistance | Would have Difficulty | Can't Do | Mean |
|---|---|---|---|---|---|
| Assigned Rating | 1 | 2 | 3 | 4 | |
| Create Geometry | 46 | 51 | 2 | 0 | 1.56 |
| Create Support | 37 | 59 | 5 | 0 | 1.68 |
| Create Load | 29 | 68 | 2 | 0 | 1.73 |
| Mesh Bar | 34 | 63 | 2 | 0 | 1.68 |

Table 7. Student Self-Assessment of Understanding Analysis Results Skills after Tutorial 1

| Skill | No Problem | Could Do w/ Tutorial Assistance | Would have Difficulty | Can't Do | Mean |
|---|---|---|---|---|---|
| Assigned Rating | 1 | 2 | 3 | 4 | |
| View Stress | 22 | 73 | 5 | 0 | 1.83 |
| View Deformation | 29 | 61 | 10 | 0 | 1.8 |

Table 8 summarizes the student self-assessment ratings broken out by Modeling-related and Results-related categories for all five tutorials. The self-assessment rating is reported as the average of the mean reported for each individual task listed under each category from the survey.

Table 8. Student Self-Assessment of Modeling-Related and Results-Related Skills

| Tutorial | Skill | Average Mean Reported |
|---|---|---|
| 1 | Modeling-Related | 1.76 |
| | Results-Related | 1.82 |
| 2 | Modeling-Related | 1.58 |
| | Results-Related | 1.66 |
| 3 | Modeling-Related | 1.83 |
| | Results-Related | 1.89 |
| 4 | Modeling-Related | 1.56 |
| | Results-Related | 1.61 |
| 5 | Modeling-Related | 1.44 |
| | Results-Related | 1.61 |

These rating results, in which a decrease in value brings the student rating closer to a desired rating of 'No Problem', indicates that student confidence in their modeling-related skills overall increased over the semester even as the models they were asked to create grew in complexity. The student's confidence in their results-related skills also improved overall but lagged the modeling-related skill ratings. This data indicates that in future offerings of this course more information and examples should be provided in the tutorials to help students better understand how to view results correctly and how to report results, compare results between trials, and to check if the results obtained by ANSYS are correct.

The number of reported student difficulties using the ANSYS Workbench 15.0[3] program environment also decreased as the semester proceeded, as shown by student responses from the online surveys that are summarized in Table 9.

Table 9. Number of Issues Reported using ANSYS Workbench 15.0[3]

| Tutorial | Students | Total Number of Issues Reported By Students | | |
|---|---|---|---|---|
| | | Workbench Interface | DesignModeler (Geometry) | Mechanical (Analysis) |
| 1 | 7 | 4 | 4 | 4 |
| 2 | 4 | 3 | 2 | 2 |
| 3 | 3 | 3 | 0 | 0 |
| 4 | 2 | 1 | 1 | 2 |
| 5 | 0 | 0 | 0 | 0 |

The student grade average for each ANSYS homework assignment was 91.3% for HW 1; 83.2% for HW 2; 84.5% for HW 3; 85.6% for HW 4; 75.7% for HW 5; and 80.4% for HW 7. The total Recitation grade students received, which consisted of grades on ANSYS homework assignments, was only 15% of the final grade in the course. In the two sections tracked in the fall 2015 assessment, several students did not complete all of the ANSYS Homework or submitted work that showed minimal effort. This low student performance was the exception, but it should be noted that under the current implementation students can choose to not complete any ANSYS assignments and still possibly pass the course. As shown in Figure 15, the majority of the students completed the ANSYS Homework Assignments and received passing grades.
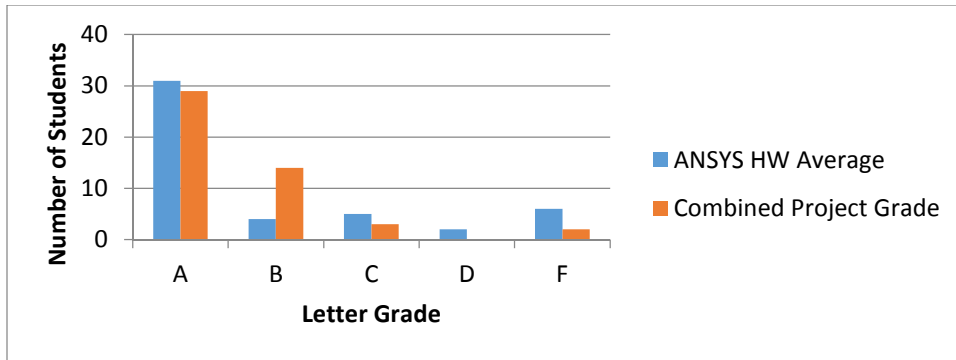


Figure 15. Student Grades Received for all ANSYS Homework and for the Combined Stress Project

Figure 15 also presents information concerning the grade distribution for the combined stress project for Fall 2015. Grades received on the Combined Stress Project were similar to grades received by students in previous semesters. For example, the final Combined Stress Project for two sections of Strengths in Spring 2008 had average grades of 85.8% (24 students) and 88.9% (18 students). For the two sections of Strengths taught in fall 2015 the average grades were 86.5% (30 students) and 87.7% (18 students).

These project averages were similar even though the Combined Stress Project in Fall 2015 required the students to successfully model and analyze the problem in ANSYS, which was not part of the project in 2008. The inclusion of the ANSYS modeling and analysis requirements for the project did not negatively affect student grades on the combined stress project. Common errors made by students on the ANSYS portion of the project included using an incorrect optimal tube thickness and failing to include a full comparison of ANSYS results with hand-calculated results. Students received on average an 88% on the portion of their grade tied to the correctness of the ANSYS model compared to an average of 86.8% on the portion of their grade tied to the correctness of the Excel spreadsheet. Based on the performance of students on the six ANSYS Homework assignments and the Combined Stress Project, the addition of substantial course material on FEA and ANSYS modeling did not significantly increase their success in the course overall as indicated by grades. However, just as importantly, the addition of forty hours of student time focused on learning to use ANSYS to the desired skill level required for the Combined Stress Project and future classes did not decrease student success in the course either. In comparing final course grades and final exam grades, the spring 2008 averages were 85% and 81% compared with the fall 2015 average final course grades and final exam grades of 84% and 79%, respectively.

**Conclusions**

Based on preliminary assessment of student performance on individual ANSYS homework assignments, online survey feedback provided by students, and student performance on a combined stress project that involved both hand calculations and ANSYS modelling and analysis, the following conclusions were made. The integration of course content using Tutorial 1 and 2 helped support student understanding of axial and bending stress and axial and bending deformation. The integration of course content using Tutorials 4 and 5 helped to support student understanding of states of stress, combined stresses, principal stresses and failure criteria. The combined stress project successfully provided students with the opportunity to show that they had developed the skill set needed to model and analyze combined load problems using ANSYS Workbench 15.0[3]. Secondary beneficial impacts of the integration of FEA content included having students investigate the effects of element types on beam deformation results in Tutorial 2 and the effect of mesh sizes and the presence of stress concentrations on stress results in Tutorial 7 and Recitation 8.

Preliminary results indicate that students acquired a sufficient understanding of the basic ANSYS modeling skills that form the foundation of subsequent courses and design project experiences, but not of basic FE theory. If the specific objective of providing students with a basic understanding of FE theory remains a primary objective in future semesters, the course framework should be changed to increase the number of weeks that FEA topics are covered in recitation sessions. Overall the integration of FEA content into the Strength of Materials course was successful in building the ANSYS skill set students needed for future classes without negatively impacting their learning of the required Strength of Material course content.

References

[1] J. Papadopoulos, C. Papadopoulos, V. Prantil, A philosophy of integrating FEA practice throughout the undergraduate ce/me curriculum, Proceedings of the 2011 Annual ASEE Conference and Exposition, 2011.
[2] T. Wedlick, S.R. Chang, B. Sepahpour. A Start Up Manual for Using ANSYS in Undergraduate Engineering Courses, Proceedings of the 2006 Annual ASEE Conference and Exposition, 2006
[3] *ANSYS® Workbench^{TM} Academic Research, Release 15.0, ANSYS, Inc., 2013*