Integration of Finite Element Software in Undergraduate Engineering Courses

John R. Baker, Vincent R. Capece, Rhonda J. Lee
University of Kentucky

Abstract

Computer-based engineering analysis tools have become more powerful and user-friendly in recent years. Most commercial software packages are now available for use on standard Windows-based PC's. Aided by increases in readily available computing power, finite element analysis (FEA) codes, in particular, have gained widespread use. FEA is now considered by many to be a standard tool for engineers. This paper outlines the incorporation of assignments based on the commercial FEA code, ANSYS, into standard lecture courses in mechanical and chemical engineering. It is now typical, at least in mechanical engineering (ME) curricula, to include course(s) specific to FEA, and these courses often include use of commercial FEA codes. Now that these codes have become more user-friendly and their plotting and animating capabilities have become more sophisticated, they can be used effectively to illustrate concepts encountered in a range of undergraduate engineering courses. The examples included in this paper are from three courses: heat transfer, fluid mechanics, and mechanical vibrations. The FEA assignments are used to complement core lecture material in the courses. They are designed so that no previous ANSYS experience, or FEA background, is required. In addition to aiding students in understanding the basic theory related to specific courses, incorporation of the software in a range of courses should increase students' awareness of the broad applicability of general-purpose FEA codes, and the advantages of using these codes as tools. It seems reasonable to expect that engineers in the 21st century will need to be comfortable with the use of FEA software and other types of analysis software in order to be effective in the workplace. Limiting student use of FEA codes to only FEA courses may not provide sufficient emphasis to the idea that these codes can be used to an engineer's advantage in many scenarios.

1. Introduction

With recent advances in computing power, commercial finite element analysis (FEA) codes have gained widespread use. FEA is now the predominant tool in stress analysis of mechanical components, and is widely used in other types of engineering analyses, such as heat transfer, fluid flow, and vibrations. While not all engineering students will become FEA practitioners, all will need a good grasp of the wide ranging capabilities, and also the limitations of FEA. Also, one of the ABET criteria is familiarity with modern engineering tools, and FEA certainly fits that category.

The fact that FEA is widely used in industry, and that engineers with FEA skills are sought in industry, is clear from a number of factors. The finite element software industry currently has annual sales in excess of $400M. The trend toward higher dependence on up-front computer-based design analysis by engineers in industry, using FEA, is documented in a recent article, *Analyzing Up Front*, in Mechanical Engineering magazine[1]. This article notes that over recent years, many analysis functions, using FEA software, have moved from engineers with Ph.D.'s, to engineers at the M.S. and B.S. degree level. Also, a recent keyword search, using the search criteria, "FEA OR FEM OR finite element", on the well-known job hunting website, monster.com[2], produced 489 job listings. Using, as a keyword, the name of the commercial code, "ANSYS" [3], the number of listings found was 124. When the name of another commercial finite element code, NASTRAN, was used alone as a keyword, there were 105 listings found.

Computational fluid dynamics (CFD) software packages based upon discretization methods, of which FEA is one, are also widely used in industry. Many engineering companies rely on CFD software as a cost-effective means to assess alternative engineering designs that depend critically on the flow behavior. It is, therefore, very beneficial to the undergraduate engineering student's employability to have obtained some exposure to a CFD software package.

FEA courses are now often included as required courses, or at least as electives, in engineering curricula (particularly in ME curricula), and these courses often include use of commercial FEA codes. For example, a recent article by Miner and Link[4], outlines the use of SDRC I-DEAS[5] software to perform static finite element analysis as part of the design process for a bracket in a Computer Aided Design course at the U.S. Naval Academy. I-DEAS is one example of a commercial code capable of performing finite element analysis. Such codes, which have become much more sophisticated and user-friendly over recent years, can now be used effectively to illustrate concepts encountered in a range of undergraduate engineering courses. Due to the well-established need for students to develop greater awareness and skills in the use of FEA, it seems reasonable to incorporate these tools into a range of courses.

This paper outlines assignments using ANSYS in standard lecture courses in mechanical and chemical engineering. The assignments complement core lecture material in three courses: heat transfer, fluid mechanics, and mechanical vibrations. They are designed so that no previous ANSYS experience, or FEA background, is required. The premise is that use of the software will not only help familiarize the students with a valuable tool, but will also assist them in learning the principles associated with the physical phenomena at the heart of these traditionally lecture-based courses. The assignments were not designed with the expectation of developing a high level of competency in using the particular FEA software chosen. In the mechanical engineering program, other courses, as outlined in the Section III, serve to better acquaint the students with specifics of FEA software use. However, there seems to be value in studying software-based solutions to standard types of problems relevant to a particular course at the time the course is taken. This is basically the same idea that is often applied when appropriate laboratory exercises are included in a lecture course. Laboratory exercises can often provide reinforcement to core lecture material more effectively than a textbook. It seems that appropriate use of finite element software, utilizing the powerful graphics capabilities of the software, can produce a similar reinforcing effect.

2. FEA Software as a Teaching Tool

Clearly, FEA is a relevant topic to engineering education. As noted above, a level of competency in FEA seems essential for engineers graduating in today's technical environment. Of course, learning to use software that produces accurate results, for example, in a heat transfer analysis, does not replace the need to understand the physical principles related to heat transfer. An enlightening discussion on the appropriate use of software in engineering courses is provided by Whiteman and Nygren[6], in which concerns are noted regarding a "black box effect". It is certainly important to emphasize to the students the need to use sound engineering judgment, and confirm results from any software analysis. Whiteman and Nygren[6], however, also point out advantages of using software tools appropriately in engineering instruction, such as making computationally intensive problems less time-consuming, and also enhancing learning, for instance, through graphics that can actually animate motion.

ANSYS has advanced graphics and animation capabilities that can aid students in visualization, for example, of fluid flow or structural vibration. In the examples outlined in this paper, graphics available within ANSYS were used to assist the students in understanding the physics of the problems they are studying. For instance, a flow field around an airfoil was animated in a fluid mechanics class, beam vibration was animated in a vibrations class, and a color contour plot showing the temperature distribution in a plate was produced in a heat transfer class. ANSYS was not used as the primary focus for these courses. However, some inclusion of ANSYS serves the dual purpose of assisting the students in learning the core material and planting a seed for future reference that a valuable tool exists which can be used to their advantage in solving a range of problems they may encounter in their careers.

While the software has become more user-friendly over the years, it is still not realistic to simply provide students a short overview of the software, and then assign a problem. Also, it would be counterproductive to expect the students to invest a significant amount of time learning to use the software in a course requiring a great deal of time and effort to learn the core principles. Care must be taken to ensure that FEA assignments in a lecture course do not produce student frustration with the software, or reduce, to any significant degree, the amount of time available for learning the basic course material. The approach taken in the assignments outlined below was to provide a full tutorial to handle relatively simple tasks in the modeling process. However, in a couple of cases, where lengthy, tedious modeling work was required, the students were supplied with macros to automate portions of the process. The ANSYS macro capability was utilized in assignments in both the fluid mechanics and vibrations courses, but for the heat transfer course, the modeling effort required was simple enough that the students could work through all modeling steps. The tutorials developed for these assignments are too lengthy for inclusion in this paper. However, student feedback from all assignments indicated that sufficient detail was provided so that the ANSYS usage portions of the assignments did not overwhelm or frustrate them.

The ANSYS work included in the courses outlined below represents an initial attempt at the University of Kentucky Extended Campus Program to better utilize ANSYS as a learning tool. The experience seemed to indicate an educational value to the assignments, based on student feedback and classroom discussions. The plan for future courses is to build on this work, and seek more open-ended design-type ANSYS projects, possibly making more use of the ANSYS macro capability. Use of macros for tedious portions of the modeling work can reduce the amount of time spent on the passive exercise of following steps in a tutorial, while still allowing inexperienced students to take advantage of the ANSYS solution and graphics capabilities as instructional aids, and raising student awareness of the wide applicability of these types of tools.

It seems that a reasonable approach is to first provide the students with a problem to solve using FEA for which there is an accurate solution they can calculate using equations, with pencil and paper. This gives the students confidence in the accuracy of the software. Then, a problem for which it would be difficult, or impossible, to obtain an accurate solution, without the software, can be provided. The more complicated problem serves to illustrate the power and usefulness of the FEA software. Also, this allows the students to "see" the physics, presented in the lecture material, in action on a system or geometry that may be more realistic than found in the types of problems to which a typical textbook is limited. This basic approach was followed, to some extent, in this first effort in the courses outlined below.

3. Instruction in FEM at the University of Kentucky

In the undergraduate mechanical engineering curriculum of the University of Kentucky, there are two required courses in which FEA is a significant component. One, *ME-501: Mechanical Design with Finite Element Methods*, focuses primarily on element formulations related to structural analysis, such as truss elements, beam elements, and 2-D planar elements, but it also includes some ANSYS use to verify results from hand calculations. Another course, *ME-406: Computer-Aided Graphics and Design*, includes instruction in solid modeling, using Pro-ENGINEER[7] software, and also use of the FEA code, ANSYS. There is a graduate-level ME course, *ME-601: Advanced CAE Applications*, which builds on ME-501, focusing largely on element formulations, and includes topics beyond static structural analysis, such as thermal and vibration analysis. ME-601 includes some ANSYS use.

Currently, the chemical engineering department does not have an undergraduate course that introduces the student to the theory of finite element analysis. Therefore, the chemical engineering student's only exposure to FEA is in the Fluid Mechanics course (see Section V), which is taken in the fall of the junior year.

4. Finite Element Applications in Heat Transfer

In the fall 2000 semester, ANSYS was introduced to mechanical engineering students in heat transfer. The project was assigned during the lectures dealing with multidimensional conduction in solid bodies. The problem is a classical one found in most textbooks in heat transfer.[8,9]

This project is illustrated in Figure 1. The problem is to determine the temperature distribution in a two dimensional solid subjected to prescribed temperatures on all boundaries. One of the desirable features of this problem is that it has a closed form solution. Hence, the students can 1) learn how to use the software, 2) compare the numerical solution with the theoretical results, and 3) through comparison with the theoretical solution learn about accuracy and the need to perform mesh refinement studies for cases where they do not know the exact solution.

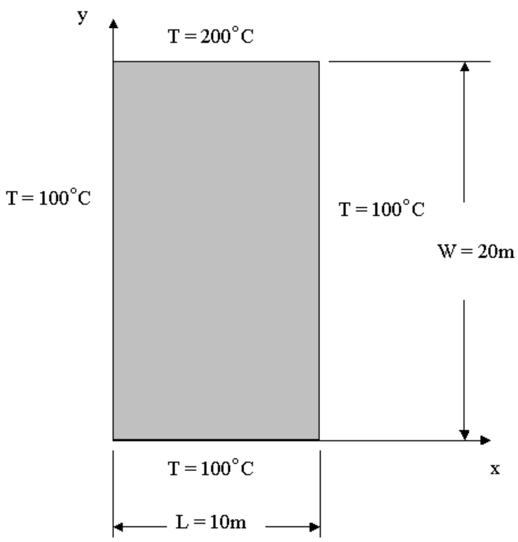

Figure 1: Conduction in a two dimensional rectangular plate.

For constant material properties the governing partial differential equation for this problem is

$$\nabla^2 T = 0,$$

which has the series solution

$$T(x,y) = 100°C + 100°C\left[\frac{2}{p}\sum_{n=1}^{\infty}\frac{(-1)^{n+1}+1}{n}\sin\left(\frac{npx}{L}\right)\frac{\sinh\left(\frac{npy}{L}\right)}{\sinh\left(\frac{npW}{L}\right)}\right].$$

Obtaining a solution using ANSYS is fairly simple using the Graphical User Interface. This involves six steps: 1) specifying the geometry to be solved, 2) defining the material properties of the solid, 3) selecting the element type, 4) specifying the mesh size, 5) solving for the temperature distribution, and 6) plotting and interrogation of the results.

For this problem the material was defined to be copper with a thermal conductivity of 401 W/m-K. A rectangular finite element (ANSYS Plane55 element, a quadrilateral element with linear interpolation polynomial) was selected for the element type. A series of meshes were used and the geometry was defined using the ANSYS preprocessing module.

Figure 2 presents the computed temperature contours for a mesh size of 24 nodes in the x-direction and 48 nodes in the y-direction using the ANSYS postprocessing module. Hence, solutions can be generated, visualized, and interrogated within the same program. To address accuracy, the students were required to perform a mesh refinement study.

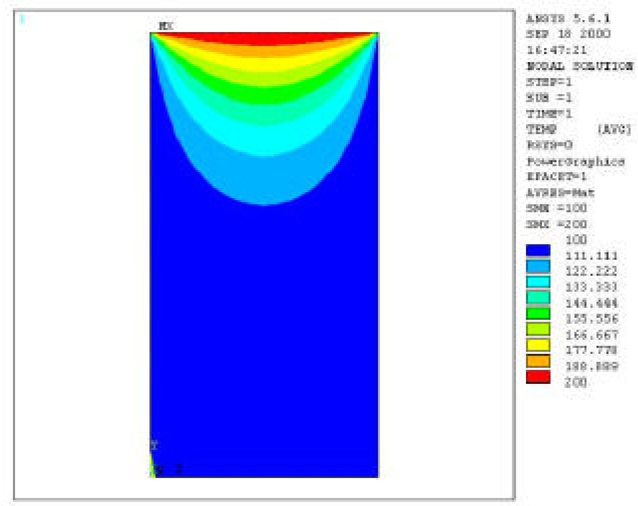

Figure 2: Computed temperature contours in a two dimensional rectangular plate.

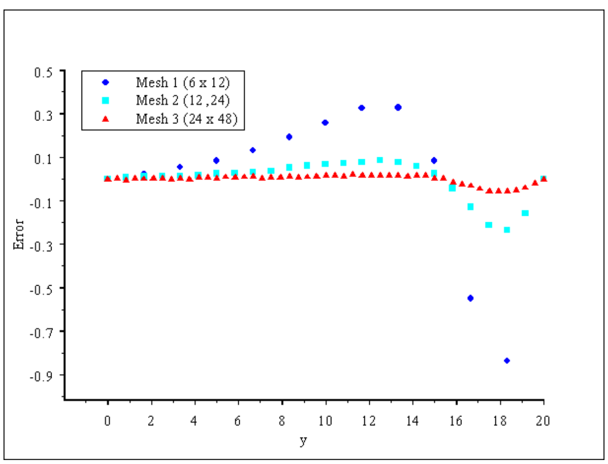

Figure 3: The influence of mesh size on the error in temperature at x = 5m in a two dimensional rectangular plate.

The error in the temperature distribution along the x = 5m plane is depicted in Figure 3. The error is defined as the theoretical temperature minus the numerical solution. The theoretical solution was obtained from the series solution given above.

Students were required to write a report on this computer project. The report included contour plots of the computed temperature within the rectangular plate, correlation of the theoretical and computed temperature distribution at x = 5m, and determination of the error in the temperature distribution at x = 5m for three different mesh sizes. The mesh sizes used were 6 x 12, 12 x 24, and 24 x 48 (where the numbers indicate the number of nodes in the x-direction followed by the number of nodes in the y-direction). Additionally, the students were required to include a discussion of the results. All students commented on the importance of the mesh size on obtaining more accurate prediction of the temperature distribution in the rectangular plate. This supported classes discussion on numerical approximation of partial differential equations for physical problems. Student comments indicated that the ability to construct the problem geometry, visualize the mesh, and visualize the solution with one program was valuable.

5. Finite Element Applications in Fluid Mechanics

Chemical and mechanical engineering students were introduced to the use of FEA to solve fluid flow problems by using a finite element method available in ANSYS 5.6 called Flotran CFD. On the day the computer project was handed out, the students were given an in-class demonstration of the ANSYS Flotran CFD software. The class example was air flowing over an airfoil, as illustrated in Figure 4. The left plot in Figure 4 shows the meshed flow field containing the airfoil, and the right plot is an ANSYS vector plot of the flow velocity, illustrating the flow separation at the airfoil leading edge. The class example was more complicated than the assigned computer project to show the students the capability of the software to predict complex flow patterns.

For the computer project, each student was given a classical flat plate, 2D, laminar, air flow problem (See Figure 5), with each student given a slightly different approach velocity (e.g. the approach velocity is 0.072764 m/s in Figure 5). The problem was adapted from the course textbook, Fundamentals of Fluid Mechanics, by Munson, Young, and Okiishi[10]. The project coincided with the introduction of the boundary layer theory (Chapter 9: Flow Over Immersed Bodies) in the text. Lecture materials included a presentation of the analytical solution or Blasius solution for flow over a flat plate.

The assignment uses the classical example of flow over a flat plate since the students can solve the same problem analytically using the Blasius solution. Since this is many of the students first exposure to CFD FEA software, one of the objectives of the assignment was to demonstrate the capabilities and limitations of numerical analysis compared with theoretical analysis to solve fluids problems. This objective was accomplished by requesting students to compare the computer solution with the Blasius solution.

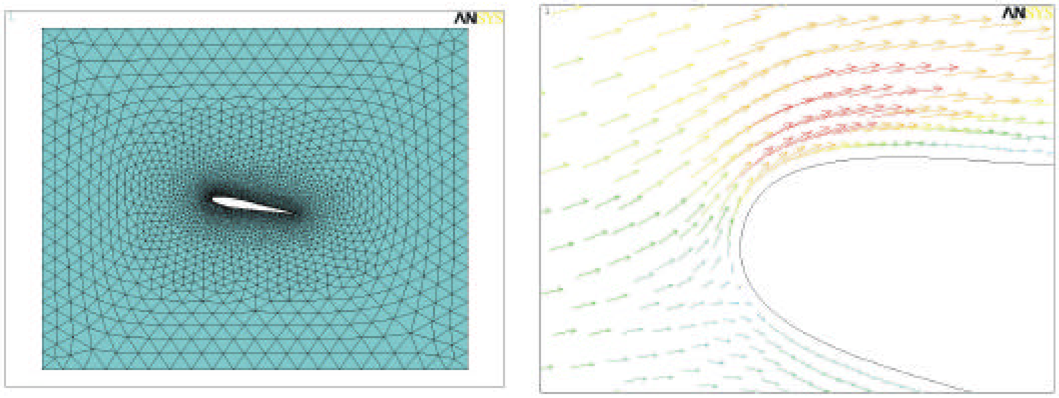

Figure 4: Airfoil – ANSYS in-class demonstration example.

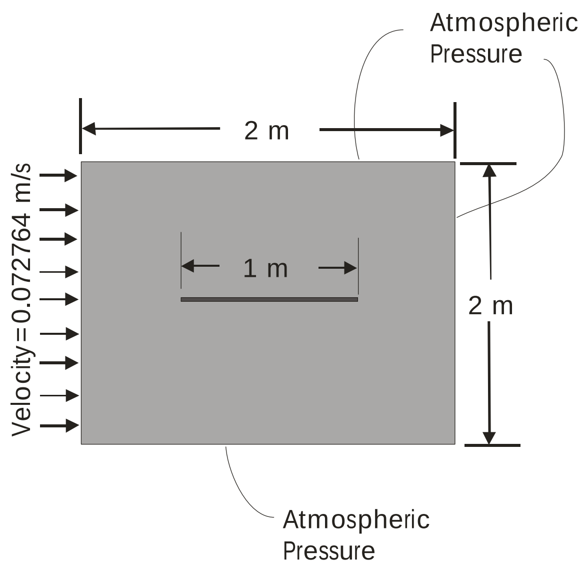

Figure 5: Set-up for fluid mechanics assignment – flow over a flat plate.

The assignment is handed out as a tutorial with detailed instructions. After launching ANSYS, the tutorial describes the required pre-processing steps, including (i) changing job name, (ii) defining element type (ANSYS Fluid141 element), (iii) defining fluid properties, and (iv) entering keypoints required to create the basic geometry (12 total). The student is prompted to save the database after entering the keypoints. Once saved, the student can resume from the point of the last save if an uncorrectable error is generated. For this reason, the student is advised to save their work often throughout the exercise. The remaining pre-processing steps are the creation of the areas, the meshing of the model, and specification of the boundary conditions and number of iterations. Once the model had been meshed, the students are prompted to zoom in on the flat plate to see the finer mesh nearer the plate. Observing the mesh illustrates for the student that a greater modeling sensitivity is required closer to the plate due to the formation of the boundary layer. After running the model with the Flotran software, post-processing steps involve listing the x-direction velocity distribution at the plate center for comparison with the analytical Blasius solution, and plotting the x-direction velocity contour plot (See Figure 6).

It was determined that two steps in the preprocessing would be better set up automatically. These steps were the entering of the grid or keypoints, and the meshing of the model. Generating the grid points is a simple procedure but easily subject to typographical errors. No value to the knowledge of the students was assumed with this step, so it was automated by using the ANSYS macro capabilities. The meshing step, which involved typing in 24 separate commands, was also automated through the generation of a macro. Therefore, for both of these steps, the student only had to enter one line of command consisting of the name of the macro. The directions for producing the mesh were provided in an attached appendix for future reference and for interested students.

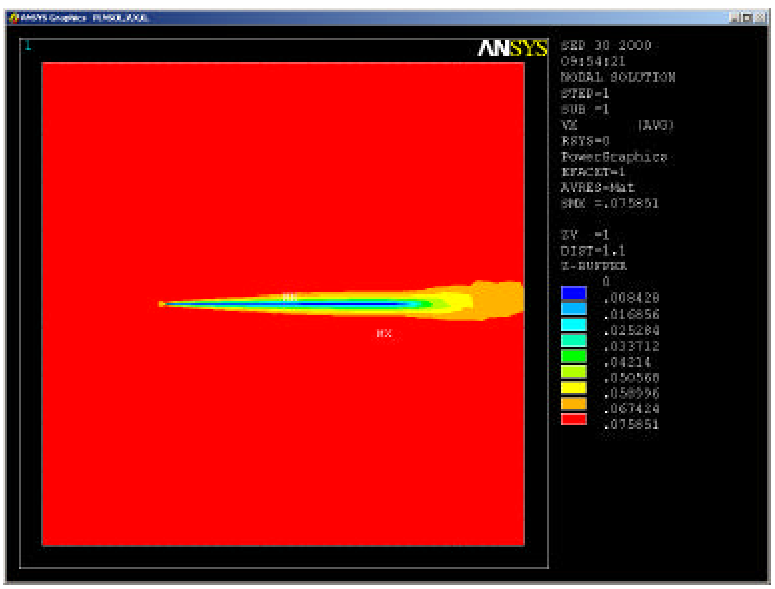

Figure 6: X-direction velocity contour plot – flow over a flat plate.

After completion of the tutorial, students were instructed to prepare a short report describing the exercise, and include a comparison of velocities predicted from the Blasius solution and ANSYS solution at the center of the plate (e.g. See Figure 7). Since every student had a different approach velocity, each student had unique results. A comparison with the Blasius solution required the students to choose values of distance from the plate, y, and calculate x-direction velocity, from tables of η (similarity variable) versus $f'(\eta)$, available in the course textbook.

Student feedback was good for the project. Many students mentioned that the comparison of the computer solution with the Blasius solution gave them confidence that these software packages could be very useful in predicting flow behavior. One surprising misconception of the students was the assumption that the computer software solution was the more accurate approach. This misconception was probably due to their lack of experience and understanding of FEA computer simulation packages.

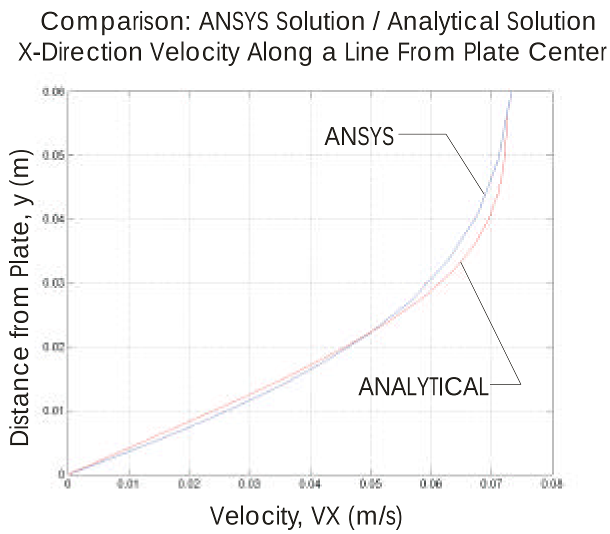

Figure 7: Example plot of comparison between the ANSYS / Blasius solutions.

6. Finite Element Applications in Vibrations

ANSYS was used in an elective vibrations course in mechanical engineering during the fall semester of 2000. Two ANSYS examples were included in the course. One was a short homework assignment, based on modal analysis of a simple two degree-of-freedom (dof) spring-mass system. The other was based on beam vibration, and was used as a basis for student presentations in a mini-project, and included both natural frequency and forced harmonic response calculations. These examples are outlined in separate subsections below.

6a) Two DOF System Vibration Example

A basic understanding of the system natural frequencies and mode shapes seems to be best accomplished by initially considering the simplest possible multi-dof system. The first ANSYS assignment was based on the simple two-dof system, illustrated in Figure 8. This system was selected as a basis for an introduction to the use of ANSYS in vibration analysis, because it is clearly one for which the system natural frequencies can be calculated by hand for comparison to the ANSYS results, and it also fits well with a simple laboratory exercise which was assigned in the course. The laboratory exercise required that students link together two laboratory masses with flexible helical springs, in

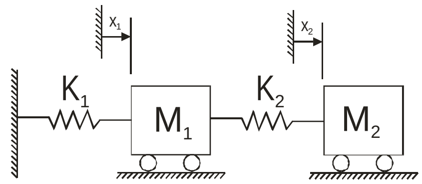

Figure 8: System for two DOF vibration example.

a configuration similar to that shown above. Students then determined the two system natural frequencies experimentally. The system had very low natural frequencies, so the students could set initial displacements corresponding to either the first or second mode shape, then simply watch, and count the oscillations, to determine natural frequencies. They also calculated the natural frequencies by hand. They then used ANSYS to analyze the very similar system in Figure 8. Because the students had observed the vibration phenomenon first-hand with an actual physical system, when the mode shapes were animated using ANSYS, the students could recognize the motion as similar to that observed for the actual physical system, and understand that the actual free motion of the physical system depended on the initial conditions. The format for this exercise was a full tutorial.

This two-dof spring-mass assignment, though a small part of the vibrations course, was intended to overview the general usefulness of FEA software as a vibration analysis tool. However, it is clear that use of FEA on such a simple system, as a practical matter, would be more of a bother than a benefit. It was clear to the students, of course, that most actual physical systems are not so simple. However, the point was made that, even though more modeling effort is required to build a finite element model of a complex system, the basic concepts related to vibration analysis using FEA are the same.

6b) Beam Vibration Example

The second FEA assignment in the vibrations class was a mini-project in which the students used ANSYS to calculate vibration characteristics for a beam shown in Figure 9. Each student was assigned one of three sets of boundary conditions for the ends of the beam: (1) clamped-clamped; (2) clamped-hinged; or (3) hinged-hinged (simply supported). Each was given an initial beam cross-sectional area, length, and material. They were told a device, with a rotating shaft, was mounted over an area at the beam mid-span, as shown. This device had some amount of mass, and produced possible harmonic excitations, perpendicular to the beam axis, at two specified frequencies, at the device mounting locations. The device mass and excitation frequencies differed for each student. Initially, they were asked to ignore the mass of the device, and calculate the first three beam bending natural frequencies using ANSYS, and compare to hand calculations based on the solution of the Euler equation for beams, from the course textbook[11]. Then, they added the mass, and re-calculated the first three natural frequencies using ANSYS. The textbook solution did not allow for the inclusion of the mass, which was not lumped at a single point, but distributed over a small region. They were then asked to perform

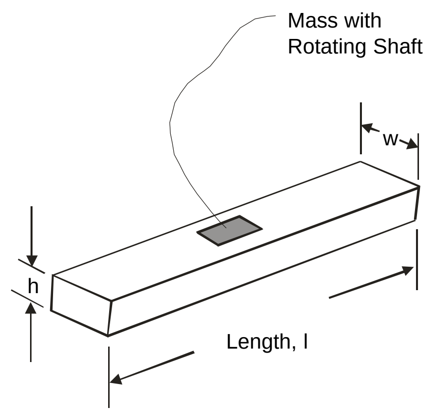

Figure 9: System for beam vibration example.

harmonic response calculations to determine the steady-state beam response amplitude, in the excitation direction, at the beam mid-span, for a frequency range including the two possible excitation frequencies. The assignment was set up so each student would find a resonance condition for their initial beam design. They were then asked to make a beam design modification to eliminate this problem. In other words, they needed to modify the beam to shift its natural frequencies so that there was no natural frequency within 15% of one of their assigned excitation frequencies.

Of course, the most efficient way to produce a beam model with ANSYS is through use of two node beam elements (ANSYS Beam3 or Beam4 elements). However, with a primary goal of the mini-project being instruction in vibration theory, there is an advantage to using the 3-D solid elements (ANSYS Solid45 elements), because then the ANSYS plots show a structure that actually looks like a beam. An additional advantage of the full 3-D model was that it allowed for graphically zooming in, at the ends of the beam, during mode shape animation, to clearly illustrate the different effects of clamping and hinging the ends of the beam. The full 3-D model also had another added advantage, in that it allowed for class discussion regarding one reason for small discrepancies between the calculated natural frequencies with ANSYS, and those calculated based on Euler beam theory, even when the added mass was initially neglected. The element formulation for the solid finite elements, of course, is not based on Euler beam theory, so convergence of the FEA results exactly to the textbook beam natural frequency results, would not be expected. Also, finite element mesh density affects solution accuracy, and this issue was discussed in class. An element plot is shown in Figure 10, upper left. Also, the second mode shape for the clamped-hinged case is shown, upper right. A zoomed view of the mode shape for the clamped end is shown, lower left, and a zoomed view of the hinged end is shown, lower right. This graphical depiction is useful in a classroom presentation regarding boundary conditions. During student presentations, ANSYS animations were presented using an LCD projector, connected to a computer.
There were a number of other aspects of note regarding the project:

- The modeling effort in ANSYS was mostly automated. The students were supplied with a macro to easily perform modeling tasks. After the first simple spring-mass assignment, the students had a basic understanding of the basic ideas of finite element modeling. So, in this assignment, it seemed that the utility of ANSYS as a tool could be illustrated without a great deal of modeling effort on the part of the students. They could quickly modify their beam design using the macro in an attempt to shift the natural frequencies, with minimal modeling effort.
- The project was assigned at the time of study of vibration of continuous systems. It seems to be slightly difficult for some students to understand mode shapes for actual continuous system. The mode shape animation capabilities of ANSYS seemed extremely useful in assisting students in understanding the meaning of the solutions of the partial differential equations in the textbook. A snapshot of an animated mode shape, from the ANSYS model, is shown in Figure 10. Of course, the finite element model is, itself, a discretized approximation to an actual continuous system. However, with a reasonably fine mesh, as in the model in Figure 10, the animated motion approximates quite well that which would be calculated from closed-form solutions for the continuous system, with the added mass neglected.
- The small amount of added mass, which was considerably less than the total beam mass, and lumped over a few nodes on the top of the beam, made the point that, in the real-world, there are complicating factors that can often be handled using a tool such as ANSYS, but it would be difficult, if not impossible, to

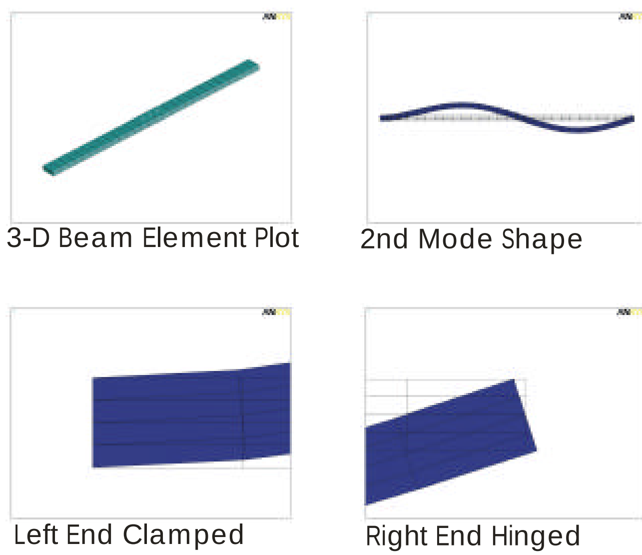

Figure 10: ANSYS plots from the beam vibration assignment.

account for these factors accurately using closed-form solutions from a textbook. Also, this added mass helped to reinforce the idea that, in general, adding mass reduces system natural frequencies.

- Some students had symmetric boundary conditions (clamed-clamped or hinged-hinged), while others had non-symmetric boundary conditions (clamped-hinged). The forced harmonic response calculations were based on excitation that was applied symmetrically about the center of the beam, very near the center of the beam. The students with symmetric boundary conditions noted that there was negligible response amplification due to excitation at the second mode frequency in the forced response analysis. This was to be expected due to the mode shape of the second mode, which shows approximately zero motion at the location of the excitation. This allowed for meaningful class discussion, at the time of the student presentations, regarding the fact that harmonic input at a system natural frequency does not produce response amplification under some circumstances. The results for the symmetric boundary conditions were contrasted with those for the non-symmetric boundary condition cases, in which some response amplification was noted corresponding to excitation at the second mode natural frequency.

7. Summary

An overview has been presented of an initial effort at the University of Kentucky Extended Campus Program to include use of a commercial FEA code, ANSYS, in three lecture courses. There are two primary purposes: 1) to assist the students in understanding the core lecture material, and 2) to raise student awareness of the wide range of capabilities of the FEA software. Since FEA software is more powerful today and is widely used in industry, it seems there is benefit to increasing student exposure to it. However, care must be taken so that it is used to enhance learning of the core lecture material, and does not hinder learning of the core material by frustrating students and draining their valuable available time. This seems to have been accomplished through use of detailed tutorials, and in some cases, automation of portions of the modeling effort through use of the ANSYS macro capability.

Student feedback, and classroom discussions, seemed to indicate that the assignments achieved the stated purposes. Future assignments, possibly with more open-ended design type problems, may provide a greater enhancement to the learning experience in a umber of standard lecture courses. A better method for gauging the success of the assignments in the future would be desirable. This could be accomplished by having two sections of the same course. One section would have ANSYS projects and the other section would have conventional assignments. Each section could then be given the same exam questions to assess command of the course material.

The fact that the same software can be used to solve problems in a wide range of courses is a benefit, because over the course of a four-year program, the students can gain some level of comfort with it. There could be concerns regarding a “black box effect”, as discussed by Whiteman and Nygren[6], if there is not a course in the curriculum on the theory of FEA. However, typical ME curricula now provide instruction in FEA theory for all students before graduation. Typically, chemical engineering students do not get exposure to the theory of FEA until the graduate level. However, exposure to the software at the undergraduate level is still considered beneficial to the student as a learning and visualization tool.

Bibliography

1. Thilmany, J., Analyzing up front, *Mechanical Engineering*, Vol. 122, No. 10, October, 2000.
2. URL: http://www.monster.com/; Job search performed under: "Search Jobs"
3. ANSYS, Release 5.6, SAS IP, 1999.
4. Miner, S. & Link, R., A Project-Based Introduction to the Finite Element Method. *Computers in Education Journal*, Vol. 10, No. 3, 2000.
5. Lawry, M., I-DEAS Master Series, Student Guide, Structural Dynamics Research Corporation, 1998.
6. Whiteman, W. & Nygren, K.P., Achieving the Right Balance: Properly Integrating Mathematical Software Packages into Engineering Education, *Journal of Engineering Education*, Vol. 89, No. 3, July, 2000.
7. Pro/ENGINEER, Release 2000i, Parametric Technology Corporation, 1999.
8. Incropera, F. and DeWitt, D., Fundamentals of Heat and Mass Transfer, Fourth Edition, John Wiley & Sons, Inc., 1996.
9. Bejan,A., Heat Transfer, Fourth Edition, John Wiley & Sons, Inc., 1993.
10. Munson, B., Young, D. & Okiishi, T., Fundamentals of Fluid Mechanics, Third Edition, John Wiley and Sons, Inc., 1998.
11. Thomson, W. & Dahleh, M., Theory of Vibration with Applications, Prentice-Hall, Inc., 1998.

JOHN BAKER

John Baker is an Assistant Professor of Mechanical Engineering at the University of Kentucky Extended Campus Program in Paducah, KY. He received his B.S., M.S., and Ph.D. in Mechanical Engineering from the University of Kentucky in Lexington, KY. After obtaining his B.S., he spent three years working in the Plastics Division of Eastman Chemical Products, Inc. He entered his current position in July 2000

VINCENT CAPECE

Vincent R. Capece is an Assistant Professor of Mechanical Engineering at the University of Kentucky Extended Campus Program in Paducah, KY. Dr. Capece received his B.S. degree in Mechanical Engineering from Tennessee Technological University, M.S. in Mechanical Engineering from MIT, and Ph.D. from Purdue University. He has held his current position since July 1999.

RHONDA LEE

Rhonda Lee is an Assistant Professor of Chemical Engineering at the University of Kentucky Extended Campus Program in Paducah, KY. She received her B.S. from Kansas State University in Manhattan, KS, and M.S. and Ph.D. degrees from The Ohio State University in Columbus, OH. She also holds an M.S. degree in Metallurgical Engineering from the University of Oklahoma. Dr. Lee was employed by International Paper Company for six years in the Environmental Technology Department. She has held her current position since July 2000.