An Effective Method of Teaching the Undergraduate Finite Element Analysis Course

ABSTRACT

The Finite Element Analysis (FEA) has become a powerful numerical method for solving complicated problems in engineering in the industry. Nowadays, in most schools the finite element course in the undergraduate curricula is taught with a mix of theory, hand-based solutions, and the use of FEA commercial software. The need to expose students at our institution (Baker College of Flint, Michigan) to the use of commercial software has been even more important since the school is mainly a career oriented institution with a strong focus on the employment of our graduates, reflected in an employment rate of 97%. While we prepare students in the use of a commercial FEA Software, extreme care must be taken to ensure that students do not compromise on the firm background of hand-based solutions. Several industries, especially aerospace, place a strong emphasis on hiring engineers with good blend of hand-based and FEA solutions. This paper discusses an effective method employed in teaching the undergraduate introductory FEA Course at Baker College of Flint. The commercial FEA software ANSYS was used in the course for the first time in the institution. The primary aim of employing the effective method was to better prepare students for advanced FEA courses, internships, senior design projects, and employment. The effectiveness of the method is assessed through a student feedback survey.

Introduction

FEA today has become an indispensable tool in many industries because of the advent of high speed computers and increased storage capacity. Consequently there has been an increased growth in job opportunities related to FEA. Many universities use commercial software in FEA courses thus providing students an opportunity to learn the method as well as use the software. Because an exposure to the finite element method and commercial software in just one FEA course is inadequate, many educators[1,2] have integrated FEA into several courses across the curriculum. At our institution, such an integration was developed in the fall quarter of last year for implementing in four courses, in sequence, in the mechanics area of the Mechanical Engineering (ME) curricula. The first of the four courses that used the integration was Solid Mechanics (ME 401). However, students who took the FEA course (ME305) last quarter had no prior exposure to the commercial FEA software ANSYS. The FEA integrations in some schools have also involved the introduction of the theory of finite element method[3,4]. The advantages of using a commercial software like ANSYS is the ease in which loads and boundary conditions can be applied and changed when necessary, to solve problems as well as the visualization of results through superior graphics and animations of deflections and stresses, which otherwise are difficult in hand-based solutions.

FEA involves preprocessing, processing, and postprocessing, all of which can be performed using ANSYS. Preprocessing involves building models, meshing, assigning material and cross-sectional properties, and applying loads and boundary conditions. Processing deals with the solution of the problem. Postprocessing involves the outputs of solutions of reactions, deflections, velocities, acceleration, stresses, as well as visualization and animation of the results. While the use of commercial FEA software has many advantages, there is a likelihood of students compromising on a firm understanding of the finite element theory and hand-based solutions. Additionally, mere using of the software can also result in students not - having a physical feel for the problem, anticipating results prior to the solution process, and performing necessary checks to verify and interpret the FEA results. Although the integration of FEA software in many early courses leading up to the finite element course is the right approach, the concern is that it instils in students taking the introductory finite element course a greater desire to use the software as opposed to learning the theory of the finite element method and hand-based solutions. This is because students already are familiar with using the software and also have the speed and confidence in using it. Despite the advantage of positioning students to work on involved projects in the FEA course due to the integration, we feel there must be a good blend of FE theory, hand-based solutions, emphasis on understanding the physics of the problem, and projects using the software.

An effective method incorporating these is discussed in this paper. The method involves the three sequential operation phases- pre-FEA, FEA, and post-FEA. The pre-FEA phase does not correspond to the typical preprocessing (FE modelling, defining properties, meshing, applying constraints and load,). Instead it pertains to the understanding of the physics of the problem, anticipation of results through inspection and calculations on equivalent simple models, minor approximations to the solution processes, and taking advantage of symmetry in the structure, all of these done prior to FEA. The next phase involves the use of FEA by both hand-based and ANSYS solution techniques. The post-FEA phase involves processing the FEA results through checking the support reactions, verifying any pre-FEA results, comparing hand-based and ANSYS solutions, interpreting results, and visualization and plotting the results. A typical FEA course will include phases 2 and 3. However, in this effective method, significant importance is given to the pre-FEA phase in an effort to inculcate in students the importance of developing a feel for the problem, engineering judgment, simple models, approximate solution for verification purposes, and anticipating results prior to FEA. In the present paper the discussion on FEA and post-FEA phases are combined for brevity, since the FEA phase involves many hand-based calculations and ANSYS commands.

The FEA course (ME305) covered topics in the areas of solid mechanics, heat transfer, and fluid mechanics. However, the problems chosen in this paper are from solid mechanics area and from the textbook[5] used for the class. The author of this textbook has done an excellent job in describing the theory and application of FEA. Three tutorials that included the analysis of beams, frames, and plane stress applications by ANSYS were developed. ANSYS was not introduced until the second half of the course in order to ensure students had a good understanding of FEA theory and hand-based solutions before using ANSYS. Students had no prior exposure of ANSYS.

1. Analysis of structures with spring elements

Any introductory FE course begins with the development of the stiffness matrix of a spring element. The stiffness matrix is derived using displacement function and equilibrium equations for nodal forces. The spring elements are then combined to represent the structure, and the structural equilibrium equations are solved.

Two spring structures analyzed are shown in Figure 1– (a) springs in series; and (b) springs in parallel. Prior to performing FEA - hand-based or software, students must be taught how to perform the pre-FEA phase.

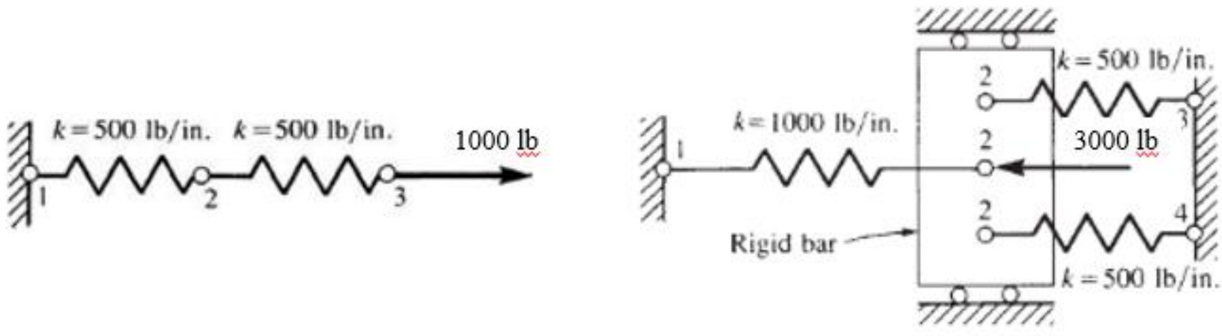

Figure 1– (a) springs in series; and (b) springs in parallel

(a) Springs in Series:

Pre FEA: The effective method used is first discussed for the structure shown in (Fig. 1a). Students were asked the following questions:

(1) Draw the structure and show the degrees of freedom (dof) on it.
(2) Draw the Free Body Diagrams (FBD) of the whole structure and isolated individual members.
(3) Identify which springs are in tension or compression.
(4) Which node will have the maximum displacement?
(5) What is the displacement of node 2?
(6) What are the magnitude and direction of the support reaction at node 1?
(7) Can the force in each spring be determined by inspection?
(8) If we know the force in each spring, can we determine the nodal displacements by inspection?
(9) Can the structure be replaced by a single spring with an equivalent stiffness k_{eq}? If so, what are the stiffness, displacement and force in the equivalent spring?

The *answers* to these are:

(1) Each node has one dof in the x (horizontal) direction.
(2) The FBD of the whole structure will include the load at node 3 and the reaction at node 1.
(3) Both springs are in tension.
(4) Node 3 will have the maximum displacement.
(5) The displacement of node 2 is one-half the displacement of node 3 because- node 1 is fixed, the two springs have the same stiffness, and the load is only applied at node 3.
(6) Due to the equilibrium of the structure the support reaction at the node 1 is equal and opposite to the applied load of F = 1000 lb.
(7) Yes. Since the applied load is 1000 lb tension at node 3, the force in the right spring from its FBD (Free Body Diagram) will be 1000 lb tension. The structure being symmetrical, the force in the left spring will also be equal to 1000 lb tension. Alternatively, the same result can be obtained since the force in the spring will be opposite to the reaction at node 1, or from the equilibrium of node 2.
(8) Yes. The displacement in the left spring is F/K = 1000/500 = 2 in. This is the relative displacement between nodes 1 and 2. As node 1 is fixed, the displacement in node 2 is 2 in. Similarly, the relative displacement between nodes 2 and 3 (or due to symmetry) is the displacement in the right spring = F/K = 1000/500 = 2 in. Since the displacement in node 2 is 2 in., the displacement in node 3 is 4 in.

(9) Yes. The structure can be replaced by a single spring with an equivalent stiffness keq. The stiffness, displacement and force in the equivalent spring are calculated as follows.

The equivalent spring stiffness for springs in series can be obtained from $1/K_{eq} = \sum (1/K_i)$, where Ki is the stiffness in a spring element i.

$$1/K_{eq} = 1/K_1 + 1/K_2$$

$$K_{eq} = (K_1 \times K_2) / (K_1 + K_2).$$

Substituting the stiffnesses of the two springs, we get Keq = 250 lb/in.

The displacement in this spring = F/Keq, which should be the displacement at node 3 in the original structure.
So, the displacement at node 3 = 1000/250 = 4 in.

The displacement at node 2 = (1/2) x 4 = 2 in.

The force in the left spring = stiffness x displacement at node 2. This results in a force of 1000 lb.

Similarly the force in the right spring must be equal to its stiffness multiplied by the difference in the displacements at node 3 and node 2 (relative displacement). This also results in a force of 1000 lb.

<u>FEA:</u> The hand-based FEA was carried out by assembling the two element stiffness matrices for obtaining the structural stiffness matrix. The resulting simultaneous equations were solved, and the answers agreed with those of pre-FEA.

(b) Springs in parallel:

<u>Pre FEA:</u> The structure shown in Fig. 1b has three springs in parallel. Many pre-FEA questions of the previous spring structure (Fig. 1a) are applicable to this structure. Spring 1 is in compression, and springs 2 and 3 are in tension. The displacement at nodes 1, 3, and 4 are fixed and therefore zero. Due to the symmetry of the structure, the support reactions at nodes 3 and 4, and forces in springs 2 and 3 are equal. By inspection, the support reaction at the nodes 1, 3 and 4 will be equal to the displacement of node 2 multiplied by the stiffness of the springs connected at their respective nodes. The reactions must all add up and be opposite to the applied load of 4000 lb. This results in,

displacement of node 2 x (K1 + K2 + K3) = 4000

displacement of node 2 x (3000 + 500 +500) = 4000.

Solving, the displacement of node 2 = 4000/(4000) = 1 in.

Alternatively, the same solution can be obtained by considering the force equilibrium of the rigid bar.

The force in each spring is stiffness multiplied by displacement of node 2. This results in a 3000 lb compressive force in the left spring and 500 lb tensile force each in the other two springs.

Alternatively, the structure can be replaced by a single spring with a stiffness equal to K_{eq}. The equivalent spring stiffness for springs in parrallel where,

$K_{eq} = \sum Ki$, where Ki is the stiffness in a spring element i.

For this example the formula results in K_{eq} = K1 + K2 + K3.

Substituting we get K_{eq} = 4000 lb/in.

The displacement in this equivalent spring = F/Keq, and should be the displacement at node 2 in the original structure. Substituting, we get the displacement at node 2 = 1 in., (=4000/ 4000) acting to the left.

The force in each spring will be its stiffness multiplied by the displacement at node 2. Thus compressive force in spring1 = 4000 lb (=4000 x 1). Similarly, the tensile force in spring 2 must be = 500 lb. (=500 x 1). Due to symmetry, the force in spring 3 will be equal to that of spring 2.

2. Analysis of structures with truss elements

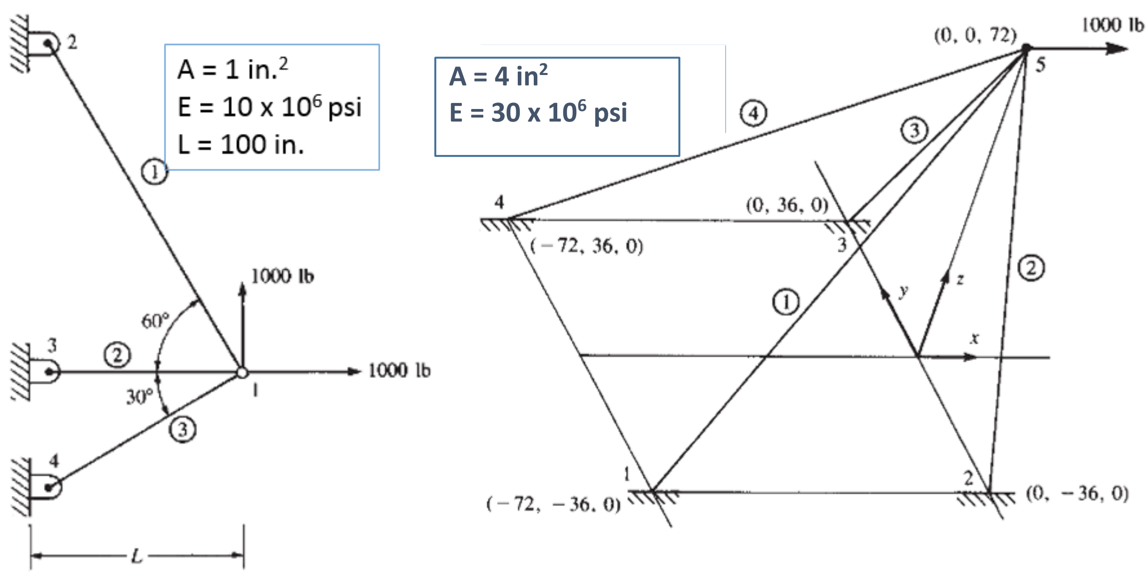

Figure 2- (a) Plane Truss; and (b) Space Truss

A two dimensional truss (plane truss, 2D) and a three dimensional truss (space truss, 3D) shown in Fig. 2a and Fig. 2b, respectively were analyzed. The derivation and the form of the stiffness matrix of a truss element is similar to that of a spring element in the local axes. In fact a truss element can be replaced by a spring with an equivalent stiffness equal to the ratio of axial rigidity

(EA) divided by the length of the element (L). However, the elemental stiffness matrices, due to the arbitrary orientations of the elements, need to be transferred to a common frame of reference (global axes) prior to assembling and solving. The truss shown in figure 3 was analyzed using both hand-based and FEA software.

Pre FEA: The following were the questions asked:

(1) Draw the structure and show the dof on it.
(2) Draw the FBD of the structure and show all loads and unsolved reactions.
(3) Which displacement will be maximum?
(4) Draw the possible deflected shape of the truss.
(5) Identify tension and compression members.
(6) Can the forces in the members be obtained by joint equilibrium?.

The *answers* are:

(1) Each node has two dof freedom (x and y) in the plane truss; and three dof (x,y,z) in the space truss.
(2) The FBD of the whole structure will have the applied loads and reactions (x and y in 2D; x, y, and z in 3D truss), assumed in the positive directions at all other nodes.
(3) The 'y' displacement will be maximum for the plane truss since member 1 is slender. The 'x' displacement of node 5 will be greater for the space truss.
(4) The deflected shape is plotted using the previous answer.
(5) For the plane truss, members 2 and 3 will be in tension; and member 1 in compression. For the space truss, members 1 and 4 will be in tension, and members 2 and 3 in compression
(6) Since joint 1 in the plane truss has three unknown forces, the forces cannot be determined by two equations equilibrium at the joint. The space truss is symmetrical and therefore forces in members 1 and 2 will be equal to 4 and 3 respectively. Thus there are only two unknown forces which can be determined by the equilibrium of joint 5. In fact only one-half of the space truss need be analyzed for FEA.

FEA and Post-FEA: Hand-based analysis was performed first. The results were identical to those anticipated in the pre-FEA. The equilibrium of the whole structure was checked. The member forces were verified by using the equilibrium of the joint. Then, ANSYS was used to solve the two trusses. The displacements and stresses obtained from the hand solution were verified through the results from ANSYS. The deflected shape was plotted, and the displacement and stress contours were visualized.

3. Analysis of Structures with frame elements

The plane frames with rigid joints were analyzed after the topic on beams were discussed. The stiffness matrix of a frame element is the combination of truss and beam stiffness matrices. The plane frame shown in Fig. 3 was considered. This frame represents a single story, single bay building, and is consists of a beam supported by two columns. The properties are: E = 30 x 10^6 psi; A = 10 in^2; I for columns (elements 1 and 3) = 200 in^4; I for beam (element 2) = 100 in^4.

The frame with loads, dimensions, and supports are shown in Fig. 3a, the dof considering the axial deformations in Fig. 3b, and dof neglecting the axial deformations in Fig.3c.

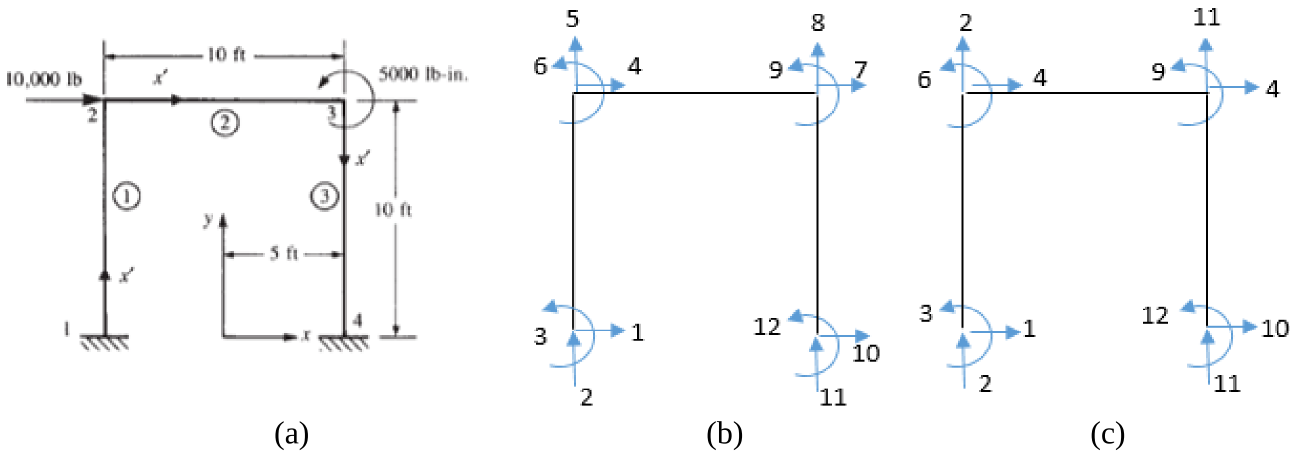

Figure 3: Plane Frame

<u>Pre FEA:</u> Almost all textbooks on FEA analyze frames considering axial deformations. However, for orthogonal frames (like in Fig. 3) the axial deformations are small and can be ignored. This significantly reduces the quantity of work in the hand-based solution.

Students were asked the following questions:

(1) Draw the structure and show the degrees of freedom on it considering the axial deformations.
(2) Draw the structure and show the degrees of freedom on it neglecting the axial deformations.
(3) Draw the possible deflected shape.
(4) Show all loads and reactions on the FBD of the structure.
(5) Draw the FBD of each isolated member showing all forces and moments.

The *answers* are:

(1) Each node has two translation and one rotational dof (Fig. 3b).
(2) See Fig. 3b.
(3) For Fig. 3b, node 2 will move up and toward right; and node 4 down and toward right. The rotation at nodes 2 and 3 will be clockwise (negative).
(4) Each fixed support will have two force reactions and one moment reaction.
(5) At the end of each isolated member, there will be three internal loadings – normal force, shear force, and bending moment.

<u>FEA and Post-FEA:</u> The frame was analyzed by hand-based solution considering axial deformations. The equilibrium of the whole frame was checked utilizing the solved reactions. The deflected shape was plotted and compared with the earlier drawn shape. Free body diagrams of each isolated member was drawn and the equilibrium of each member was verified using the equations of equilibrium. Axial force (N), shear force (V) and bending moment (M) diagrams for each member were plotted and combined to get the V and M diagrams of the whole frame. The axial deformation in each member was determined. The hand-based solution for displacements corresponding to the dof for the frame in Fig. 3b are shown in Table 1.

Type	Degree of freedom (dof)	Value	Units
Displacement	4	0.211	in.
Displacement	5	0.00148	in.
Displacement	7	0.209	in.
Displacement	8	-0.00148	in.
Rotation	6	-0.00153	rad.
Rotation	9	-0.00149	rad.

Table 1: Displacement and Rotation for Frame Considering Axial Deformations

The axial deformation in the beam (element 2) is the difference in the displacements at its ends (0.209 – 0.211 in), which is equal to -0.002 in. It is evident that the axial deformations in the beam (= 0.211 in) and columns (=0.00148 in) are very small when compared with the dimension of the members and the lateral sway of the frame (0.211 or 0.209 in.) and therefore can be ignored.

Next, the frame was solved using ANSYS. The results of ANSYS were used to verify the support reactions, nodal displacements, nodal rotations, member end forces and moments, obtained from hand-based solutions. The V and M diagrams (Fig 4) were plotted using ANSYS and checked with the earlier plots drawn from hand-based solution. The axial deformation in each member was determined. One of advantages of using ANSYS is the plotting of V and M diagrams. Users can clearly see the variation of shear and moment and regions of high values.

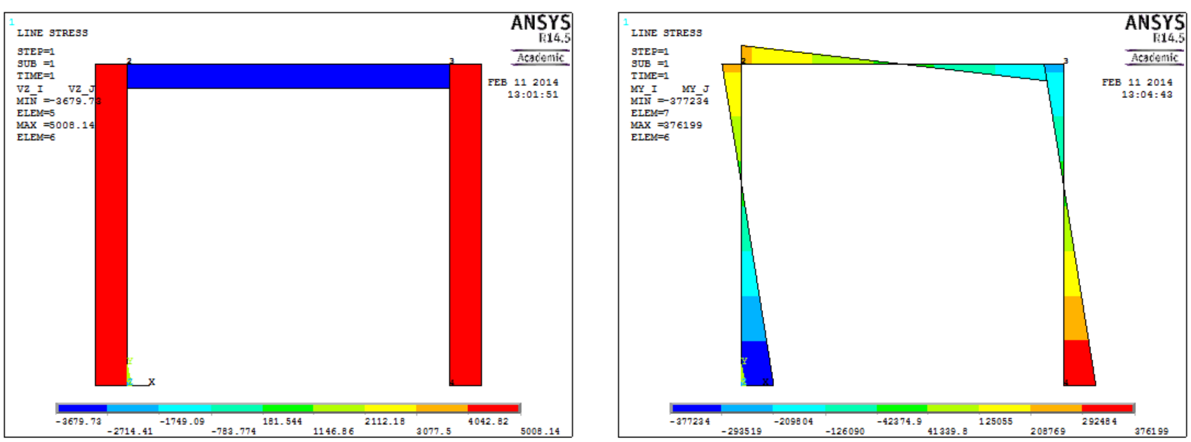

Figure 4: (a) Shear (V) diagram; (b) Bending moment diagram (M)

Finally, the frame was analyzed using hand-based solutions with the axial deformations neglected. The degrees of freedom for this case are shown in Fig. 3c. In the frame in Fig. 3b, the axial deformations are –the difference in the displacements corresponding to the degrees of freedom (dof) 4 and 7 for the beam; the displacement corresponding to dof 5 for the left column; and the displacement corresponding to dof 8 for the right column. However if the axial deformations are neglected, the end nodes of each member must have the same axial dof reference number. This means that each unconstrained member will translate axially as a rigid body. The advantage of this method, only applicable to orthogonal frames, is that the stiffness matrices need not be transformed from local to global axes, and the beam stiffness matrices can be used directly by appropriate selection of the local axes. For the columns the local x axis is chosen such that it goes from the top node to the bottom node, and for the beams from the left node to the right node. This greatly reduces the effort involved in hand-based FEA when compared with the analysis considering axial deformations. The hand-based solution for displacements corresponding to the dof for the frame in Fig. 3c are shown in Table 2.

	Degree of freedom (dof)	Value	Units
Displacement	4	0.209	in.
Rotation	6	-0.00150	rad.
Rotation	9	-0.00148	rad.

Table 2: Displacement and Rotation for Frame Considering Axial Deformations

Comparing these results with those obtained by considering axial deformations for the frame (Fig. 3b) it is very clear that the two solutions are identical, thus justifying the advantage of neglecting axial deformations in orthogonal frames.

3. Analysis of structures with plane strain elements

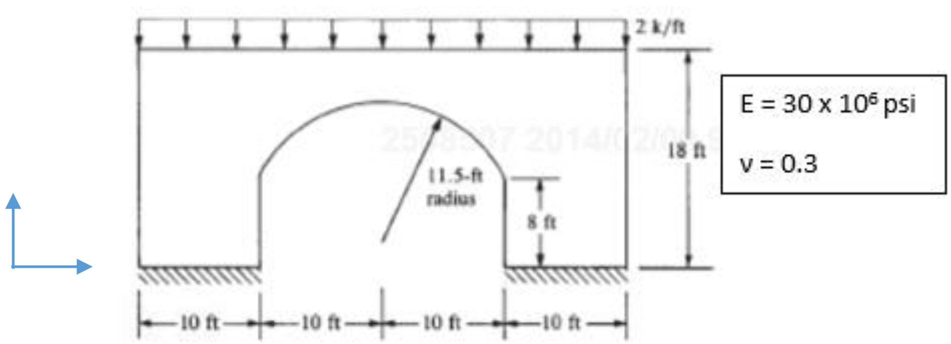

Figure 5: Concrete overpass structure

In a typical FEA course, students are taught the solution of problems related to plane stress. In the authors' opinion many students do not know the difference between plane stress and plane strain. In the FEA course (ME305), students were exposed to both plane stress and plane strain problems through hand-based FEA and ANSYS solution methods. Plane strain is basically a two dimensional deformation in which due to the third dimension being very large as compared to the other two, results in the displacements and strains, but not the stress, in the third dimension to be zero. Typical examples are dams, long cylinders, overpass structures, etc. In an effort to help students better understand the difference between plane stress and plane strain, a concrete overpass structure (Fig. 4, a plane strain problem) was assigned as a project assignment using ANSYS.

<u>Pre-FEA:</u>

Questions:

(1) Why is the concrete overpass a plane strain problem?
(2) Why is the 3D structure analyzed as a 2D problem?
(3) Would you analyze the whole structure shown or a reduced model?
(4) If a reduced model could be used what constraints would you apply?
(5) Would you use one finite element mesh to solve the problem?
(6) What is the magnitude and direction of the total support reaction?
(7) Which displacement is greater (x or y)?
(8) Which normal stress is greater (x or y)
(9) In which region are the stresses higher?
(10) Is the stress in the z direction zero?
(11) How would you ensure the safe design of the structure?

Answers:

(1) The longitudinal dimension (z) is very long and the strains in this direction very small and therefore negligible, thus making it a plane strain problem
(2) The strains are in two dimensions only.
(3) Since the overpass has one axis of symmetry, only one half of the structure was analyzed.
(4) Symmetry boundary conditions will be applied. In this case the y axis at the center is the axis of symmetry. All nodes on this axis of symmetry will have zero x-displacement.
(5) No. Several meshes will be used going from a coarser mesh to s finer.
(6) The total reaction must be equal opposite to the downward load. The downward load is 80,000 lb. (2 kip/ft x 40 ft).
(7) The displacement in the y direction will be greater than x.
(8) The stress in the y direction will be greater than x due to the downward load.
(9) The region close to the load will have higher stresses.
(10) No. The stress in z direction (unlike the z strain) is not zero.
(11) The structure is made of concrete which is weak in tension. The maximum tensile stress in the structure are obtained from the analysis and compared with the tensile strength of concrete. If the stress exceeds the tensile strength, cracks perpendicular to the direction of the maximum tensile stress will be formed. Either the design of the structure has to be modified or the structure must be reinforced with steel appropriately.

<u>FEA and Post-FEA:</u>

There was no hand-based solution for this problem. The structure was analyzed using ANSYS with several mesh densities to understand its influence on the solution. Only one half of the structure was considered due to the symmetry. All modeling was done using ANSYS. The results were verified for support reactions prior to interpreting displacements and stresses. The deformation configuration and the stress contours were plotted. Figure 6 has the contours of x and y normal stresses for the left half of the structure. The y stresses are much larger than the x stresses as anticipated in pre-FEA. The contour plots of the principal stresses which give the maximum and minimum normal stresses, and the maximum shear stresses were also visualized. The contour plot of vonMises stress which gives one single value of for all combinations of stresses was also reviewed.

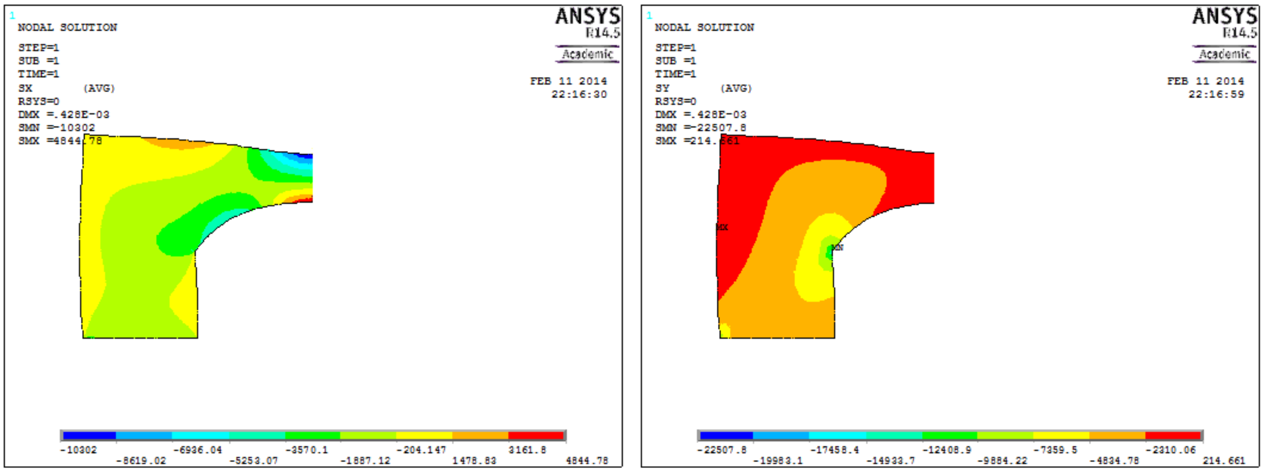

Figure 6: (a) Normal stress x (psf); (b) Normal stress y (psf)

Student Feedback

Since ANSYS was used for the first time in the FEA course and also that students had no prior exposure to ANSYS, the feedback questions were mainly aimed at analysis using ANSYS. The FEA courses should be taught using a commercial software which addresses one of ABET student outcome 'k'- Ability to use the techniques, skills, and modern engineering tools necessary for engineering practice. The class sizes at our institution name are generally small to conform to the school's requirement of lower student to faculty ratio. Furthermore, the course is offered as an elective. Therefore it is not uncommon to have few students enrolled in the course. Seven students were available to participate in the survey. The survey results must not be looked to see if it has any substantial statistical relevance but rather to check if the developed tutorials and the use of ANSYS are directionally correct. More data will be collected after the course is taught several years for an in-depth statistical analysis.

Table 3 shows the survey results for the individual tutorials. 71 % strongly agreed that the ANSYS tutorials helped them in developing speed, mastery and confidence in using FEA software. This was very positive given that students has no previous exposure to ANSYS. Questions 2 through 6 are specific to the individual tutorials, and the feedback was mainly in the "strongly agree" and "agree" categories. 57 % strongly agreed and 43 % agreed that the tutorials helped in appreciating the advantages of using ANSYS for solving large and complicated problems, and that using ANSYS in the course has increased their interest in applying and learning more about finite element analysis.

		Strongly disagree, %	Disagree, %	Neutral, %	Agree, %	Strongly agree, %
1.	The Finite Element Analysis Software ANSYS Tutorials in the ME305 course were a good learning experience.				43	57
2.	**Tutorial 1:** "Analysis of Beams" helped me understand the concept of reactions, deflections, stresses, and shear and moment diagrams.				57	43
3.	**Tutorial 2:** "Analysis of Plane Frames" helped me understand the concept of displacements, stresses, and shear and moment diagrams in 2D frames.				57	43
4.	Tutorial 2 helped me appreciate that large frame structures like multistory buildings can be easily solved using ANSYS when compared with hand solutions.				71	29
5.	**Tutorial 3:** "Plane Stress- Stress Concentration" helped me understand the use of 2D elements, stresses, and stress concentration in plane stress problems.				57	43
6.	Tutorial 3 and the projects enhanced my understanding of the influence of mesh size on the FEA.			14	29	57
7.	The ANSYS tutorials increased my theoretical knowledge of FEA			14	57	29
8.	The ANSYS tutorials helped me in appreciating the advantages of using commercial software when solving large and complicated problems.				43	57
9.	The verification of ANSYS results with hand solutions helped me appreciate both methods of solution.				57	43
10.	The use of ANSYS in this course helped me visualize the deformed shape of the structure, and contours of displacement and stresses.			14	43	43
11.	Overall the ANSYS tutorials added value to the ME305 course.			14	43	43
12.	The ANSYS tutorials helped in developing speed, mastery and confidence in using FEA software.				29	71
13.	Using ANSYS in this course has increased my interest in using and learning more about finite element analysis.				43	57
14.	I would recommend including ANSYS based hands-on Tutorials related to course topics in other courses in my program.				43	43

Table 3. Student Survey Results on the Effectiveness of the Individual Tutorials I and use of ANSYS

Conclusions

An effective method of teaching the undergraduate FEA course is presented. The method involves the use of three phases of analysis in sequence for solving problems- pre-FEA, FEA, and post-FEA. Most FEA courses are taught with less attention to pre-FEA, and sometimes even to post-FEA. The pre-FEA phase is also not emphasized or discussed in detail in the textbooks. The method discussed in this paper emphasizes the need for a pre-FEA phase for a better understanding of the physics of the problem, anticipation of results through inspection and calculations on equivalent simple models, minor approximations to the solution processes, and taking advantage of symmetry in the structure. The advantages of the method is demonstrated by considering problems solved in the FEA course that cover four topics. Furthermore, in the post-FEA phase, the verification of equilibrium prior to interpreting displacements and stresses has been overly emphasized. The authors, based on their work experience in the industry, strongly feel, that many finite element analysts do not do a pre-FEA study before analyzing nor perform equilibrium checks post-FEA prior to reviewing displacements and stresses. A good blend of FE theory, hand-based solutions, and use of ANSYS was taught in the FEA course. The ANSYS software was introduced for the first time in a FEA course at Baker College of Flint. Students did not have any exposure to ANSYS prior to taking this course. Integration of ANSYS in the ME curricula across four courses in the mechanics area, in sequence, has been initiated last fall quarter. Future students going into the FEA course will have the speed and confidence in using ANSYS due to this integration. While many real world problems may not deal with simple structures, the method discussed in this paper prepares students to become better finite element analysts for advanced FEA courses, internships, senior design projects, and employment.

References

1. Papadopoulos, J.M., Papadopoulos, C., and Prantil, V.C., “A Philosophy of Integrating FEA Practice Throughout The Undergraduate CE/ME Curriculum”, *Proceedings of the ASEE Annual Conference & Exposition*, Vancouve, BC, Canada, June 26-29, 2011.

2. Baker, J.R., Capece, V.R., and Lee, R.J., “Integration of Finite Element Software in Undergraduate Engineering Courses,” *Proceedings of the ASEE Annual Conference & Exposition*, Albuquerque, NM,, June 2001.

3. Zhao, J. “Teaching Finite Element Analysis as a Solution Method for Truss Problems in Statics”, in *Proceedings of the ASEE Annual Conference & Exposition*, Salt Lake City, UT, June 2004.

4. Chaphalkar, P. and D. Blekhman. “Introducing Finite Element Analysis in the First Course of Statics and Solid Mechanics”, in *Proceedings of the ASEE Annual Conference & Exposition*, Honolulu, HI, June 2007.

5. ANSYS, Release 14.5, 2012

6. Logan, D.L., “A First Course in the Finite Element Method”, 5th Edition, *Cengage Learning*, 2012.