Teaching Finite Element Analysis in an MET Program

Jack Zecher, P.E.
Indiana University Purdue University Indianapolis

Abstract

During the past decade finite element analysis (FEA) has transitioned from a specialized tool to one that is often used on a daily basis during the design process in industry today. This is because FEA, running on desktop computers, can solve complex problems that are impossible to solve by hand. Due to this popularity of FEA, the MET program now requires that all students take a courses in finite element analysis during their junior year. This paper outlines the material covered during this course.

Finite element analysis has proven to be a very powerful software tool that provides users with a great deal of analytical clout. However, users of finite element analysis programs must have a solid understanding of its underlying principles in order to obtain accurate results. A primary objective of this course, therefore, is to prepare students to be responsible users of finite element analysis programs. The first several weeks of the course cover the background of FEA , including the basic stiffness matrix approach using one dimensional spring elements. Modeling techniques are then introduced, that deal with topics such as: mesh size, aspect ratio, poorly shaped elements, boundary conditions, and use of symmetry. The remainder of the course deals with the use of various element types and different solution types. The majority of the course covers FEA from a stress analysis point of view, thus, reinforcing concepts from previous courses in Statics, Strength of Materials, and Machine Elements.

Format of the course is 2 hours of lecture and 2 hours of lab per week. Ten written lab report projects are assigned during the semester. Most of these lab projects consist of preparing and analyzing finite element models of parts that have known theoretical solutions. This approach gives students “theoretical benchmarks” against which they can compare their FEA results, and observe how changes to their models (such as varying the mesh size) affect their results. This technique has proven to give students confidence in using FEA to produce corrent results, while also instilling a respect for how easy it is to obtain erroneous results.

Introduction

The increased computing power of personal computers has now made finite element analysis a widespread tool use through many different industries. It is now the predominant tool in stress analysis of mechanical components, as well as being used extensively for other types of analysis types; such as: heat transfer, fluid flow, and vibration analysis. While not all MET graduates will end up being FEA practitioners, they all should understand its capabilities and also its limitations.

The course discussed in this paper, is a junior level course that focuses primarily on using finite element analysis to solve linear stress analysis problems. Prerequisites to the course include: Statics, Strength of Materials, and Design of Machine Elements. Unlike some MET programs that have chosen to introduce FEA in their Statics [1] or Strength of Materials courses [2], the course described in this paper centers on FEA as the main focus of the course. This approach allows concepts and equations developed during these earlier courses to be reviewed and reinforced by comparing their manual calculations with the results produced from the finite element model.

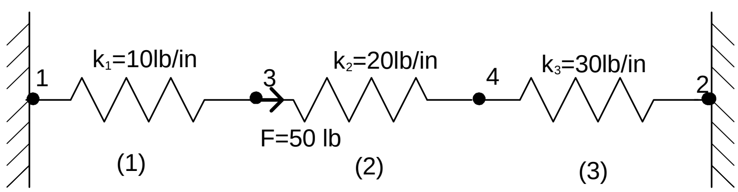
Ten written lab report projects are assigned during the semester. Most of these consist of preparing and analyzing finite element models of parts that have known theoretical solutions. This approach gives students “theoretical benchmarks” against which they can compare their FEA results, and observe how changes to their models (such as varying the mesh size) affect their results. This technique has proven to give students confidence in using FEA to produce corrent results, while also instilling a respect for how easy it is to obtain erroneous results.

This process of comparing theoretical and FEA results actually starts during the first lecture of the course, in which a finite element model of a cantilevered beam is prepared and analyzed during the lecture. The students are given 15 minutes to manually calculate the stress at three different points on the beam prior to viewing the FEA produced stress results. The manual calculations and computer results are then compared. This has proven to be an "eye opening" experience, for both the student and instructor, to find how many students have forgotten how to calculate bending stress, shear stress, as well as, how to combine these values using Mohr's circle in order to find the maximum principal stress.

This process of comparing FEA results with manual calculations is continued throughout the semester wherever possible. It provides the students with what sometimes is a much needed review of previous concepts and equations, as well as illustrating to the students to not just trust any result that is generated from a finite element analysis.

Introduction of Finite Element Theory

After spending the first week's lecture introducing finite element terms such as: nodal points, elements, degrees of freedom, boundary conditions, etc. and then comparing the FEA solution of the cantilevered beam to manually calculated values, the second week is spent on FEA theory. The amount of theoretical background that is covered in this course is quite limited as compared to finite element analysis courses that are taught in engineering programs. While engineering courses quite commonly develop the theoretical basis of each different type of element's stiffness matrix, this course limits the development of stiffness matrices to 1-dimensional spring elements. These one dimensional element stiffness matrices are then combined (using the direct stiffness method) into a global stiffness matrix that represents the entire model. Finally, the process of introducing boundary conditions (restrained nodal points) and its resulting effect on the makeup of the global stiffness matrix is discussed.



$$K = \begin{bmatrix} 10 & 0 & -10 & 0 \\ 0 & 30 & 0 & -30 \\ -10 & 0 & 10+20 & -20 \\ 0 & -30 & -20 & 20+30 \end{bmatrix}$$

Figure 1 Three element spring assemblage and its resulting stiffness matrix

The homework assignments during this portion of the course requires the student to go through the steps of manually assembling the global stiffness matrix and then solve the resulting simultaneous equations after reducing the global matrix equations where boundary conditions are applied. This process helps students to understand of how degrees of freedom at each nodal point are related to particular equations. It also give them an appreciation of how the size of the problem grows as the number of nodal points increase.

Starting to Prepare FEA data

In order to begin the process of performing a finite element analysis, the third week is spent going through the steps required to prepare and analyze a cantilevered beam. The beam is modeled using quadrilateral plane stress elements. During this process, the emphasis is on only learning the steps to go from CAD data to stress and displacement results. This involves creating the mesh and then adding boundary conditions, material properties, concentrated loads, etc. Discussion of various finite element modeling techniques that are used to produce more accurate results is delayed until subsequent projects. At this point students are instructed to build models that have square elements and a course mesh.

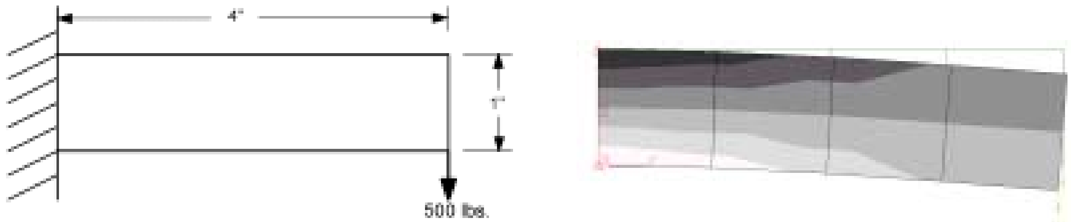


Figure 2 Example cantilered beam model discussed in first lecture

The first lab project involves performing a linear static analysis of a simply supported beam and then compares their FEA displacement and stress results with manually calculated values. The concept of beam deflection due to shear[3] is also discussed during this week. This lays the foundation for more accurate deflection comparisons in subsequent labs.

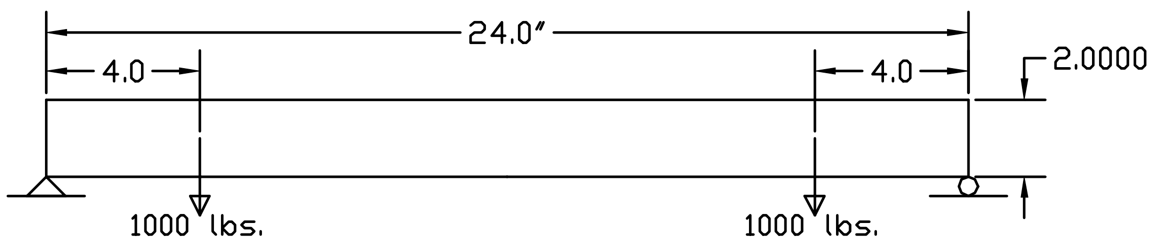


Figure 3 First Student lab project

Modeling Techniques

The importance of using proper modeling techniques to insure that a finite element analysis produces the correct results cannot be overemphasized. Competent users must understand how to select suitable types, sizes, and shapes of elements in order to prevent misrepresentation of the physical part. Since finite element analysis is a numerical approximation of the actual physical part, it is important that the user has a good understanding of both the part being analyzed as well as the limitations of the finite element process. Within this portion of the course, techniques that deal with: using different mesh sizes in areas of high stress gradients, converging to the correct result through mesh refinement, aspect ratio and badly shaped elements, assigning proper boundary conditions, how the application of concentrated loads can introduce artificial stress concentrations, and use of symmetry.

The second lab project involves analyzing a beam that is completely constrained at both ends and loaded at its center. This is done by first building an FEA model of the full beam, and then reanalyzing the beam by building a half-model that uses symmetrical boundary and loading conditions.

The third lab project makes use of the concept of using a finer mesh of elements in regions of high stress gradients. This again involves creating a finite element model of plane stress elements. The part analyzed is an axially loaded plate which contains a centrally located hole. This project also allows the FEA results to be compared to manual calculations.

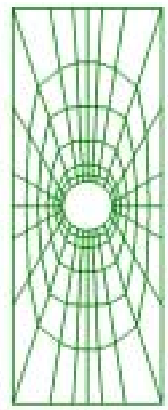


Figure 4 Example mesh illustrating smaller elements in regions of high stress concentrations

Axisymmetric Elements

Topics introduced during this portion of the courses involve how to model pressure vessels and axially loaded stepped down shafts. Illustrations of how both concentrated and pressure loads are applied are discussed. Because Algor provides a large number of meshing techniques, several of them are introduced and used during this week's project. Again, finite element results are compared to manual calculations.

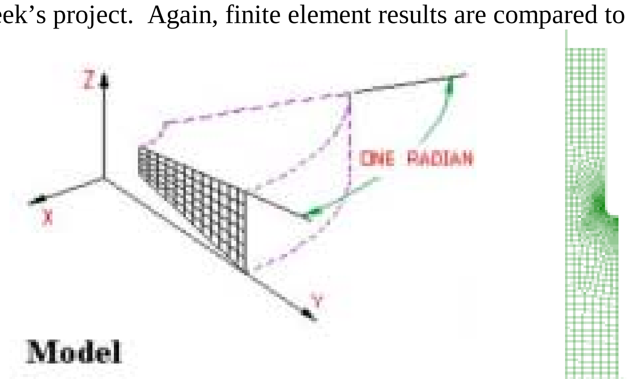


Figure 5 Axisymmetric element concept and the FEA mesh of a step down shaft

Truss and Beam element models

During the next two weeks 2-dimensional truss modeling and 2-dimensional beam modeling techniques are introduced. The truss element model is compared to results that are manually calculated using the "method of sections". Results of the beam element model, are likewise, compared to manual calculations.

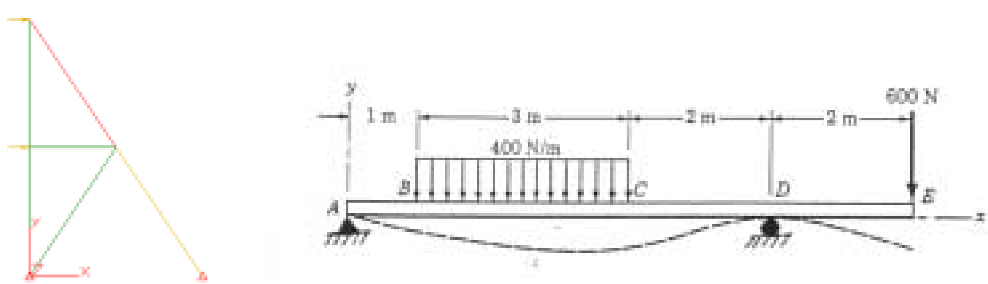


Figure 6 Truss and Beam lab projects

Remaining lab Projects

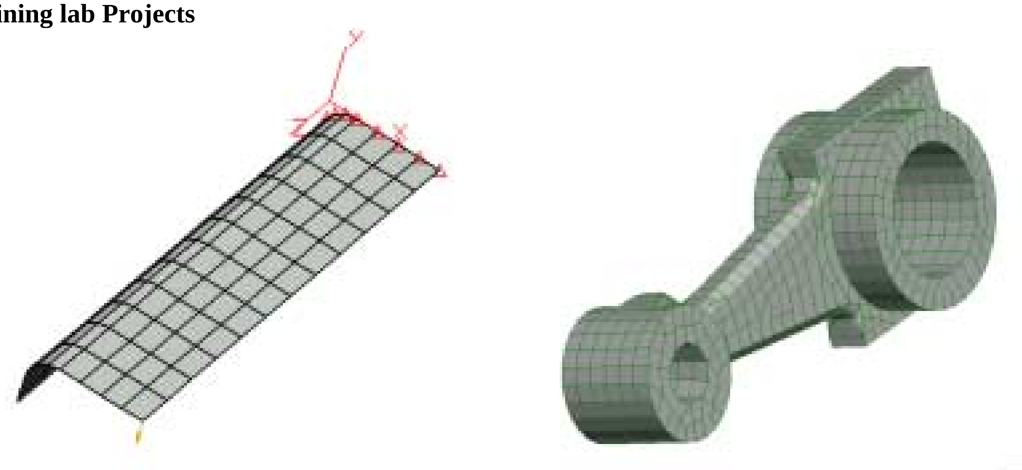


Figure 7 Plate and 3-D solid element models

During the final portion of the course, students learn how to create finite element models using plate and 3-D solid elements. The lab projects do not have direct "manual calculations" which can be compared as the former lab projects do. However, due to concepts learned in the first portion of the course, where students have these theoretical formulas to compare their FEA results to, the majority of students are able to achieve correct results in these latter lab projects.

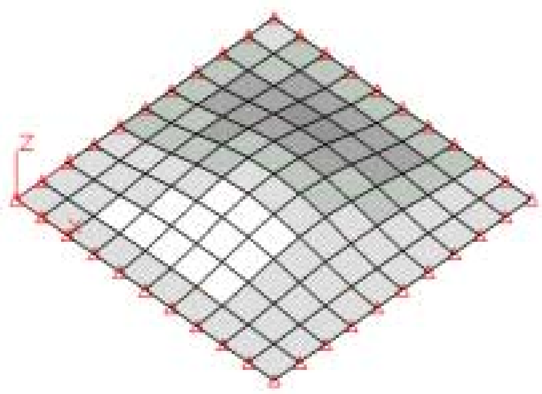


Figure 8 Natural frequency and mode shape analysis of a clamped plate

Conclusions

Users of finite element analysis programs must have a solid understanding of FEA modeling techniques in order to obtain accurate results. This paper has described one approach that is focused on that goal. Revisiting statics and stress analysis problems seen in previous courses gives students "theoretical benchmarks" against which they can compare their FEA results, and observe how changes to their models (such as varying the mesh size) affect their results. This approach has proven to give students confidence in using FEA, while also instilling a respect for how easy it is to obtain erroneous results.

References

[1] Pike, M., "Introducing Finite Element Analysis in Statics", ASEE Conference 2001

[2] Logue, L. and Hall, K., "Introducing Finite Element Analysis in an MET Strength of Materials Course", ASEE Conference 2001

[3] Popov, E.P., Mechanics of Materials, 2nd Edition, Prentice-Hall