AC 2008-1050: DEVELOPMENT OF AN INSTRUCTIONAL TUTORIAL FOR TEACHING FINITE ELEMENT ANALYSIS USING ANSYS WORKBENCH

John Zecher, Indiana University-Purdue University-Indianapolis

Development of an Instructional Tutorial for teaching Finite Element Analysis using ANSYS® Workbench™

Abstract

During the past several years, use of finite element analysis (FEA) in industry has transitioned from a specialized tool, used primarily by full-time analysts, to one that is used by product engineers as an integral part of the design process. A major reason for this shift is that new FEA software packages, such as ANSYS Workbench, employ user interfaces that are similar to CAD systems. These packages provide automatic meshing capability and allow loads and constraints to be applied directly to the three-dimensional CAD model's edges or surfaces, rather than requiring the user to work with traditional FEA objects like nodal points and elements.

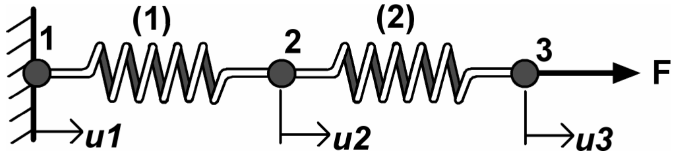
This paper discusses the implementation of ANSYS Workbench within an MET finite element analysis course. In order to effectively teach finite element concepts and the ANSYS Workbench software, an instructional tutorial book and related multimedia CD-Rom have been developed. The tutorial not only contains material normally included in user's manuals that cover only "the FEA tool", but also material covered in traditional FEA textbooks that concentrate on the mathematical development of FEA theory.

The primary objective of the tutorial is to provide instructional material that will help both students and product engineers become responsible users of the finite element analysis program. Because the FEA process used in ANSYS Workbench is so much easier than those employed by previous generation FEA programs, there is a greater possibility of having unqualified users performing analyses and generating incorrect results. In order to avoid this situation, the tutorial focuses on providing a basic understanding of the underlying principles of FEA as well as a step-by-step guide of how to use the Workbench "tool".

Introduction

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 Strength of Materials, and Machine Elements. Unlike some MET programs that have chosen to introduce FEA in their Statics or Strength of Materials courses[1,2], the course described in this paper centers on FEA as the main focus. This approach allows concepts and equations developed during earlier courses to be reviewed and reinforced by comparing manual calculations with the results produced from the finite element model.

This finite element course differs from a typical engineering FEA course from the standpoint that it emphasizes the process of building finite element models rather than being theoretically based. The course begins by introducing basic FEA terminology and then covers the basic stiffness matrix approach using one-dimensional spring elements. The objective of this portion of the course is to give students a fundamental understanding of how FEA numerical computations are done. However, within the course, this is the only coverage of developing stiffness matrices.



$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1+k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

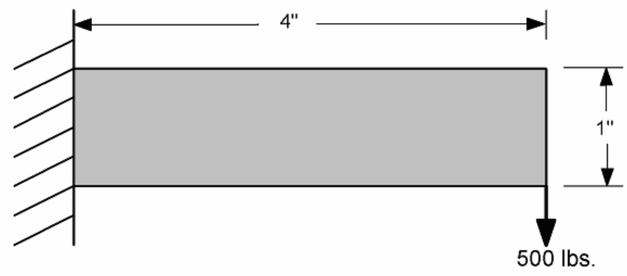
Figure 1 Typical one-dimension spring element problem with its corresponding stiffness matrix

Modeling techniques that focus on the following topics are emphasized: mesh size, convergence, aspect ratio, poorly shaped elements, boundary conditions, and the use of symmetry. The remainder of the course deals with the use of various element types and different solution types. Most of these example 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 also allows them to 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 and an understanding of how to produce correct results, while also instilling a respect for how easily erroneous results can be produced.

For the past two years the ANSYS Workbench software has been used in the course. This software is typical of an entirely new class of FEA software that allows users to work with finite element models in an environment that is similar to a CAD program. ANSYS's vision is that Workbench will be used by product engineers who will only use it occasionally as part of their job, rather than by structural analysts whose main activity is to do FEA work. Since many more MET graduates are likely to find employment opportunities as product engineers rather than as finite element analysts, it makes sense that this is the type of software that should be used in the curriculum.

Relating FEA Examples to previous course work

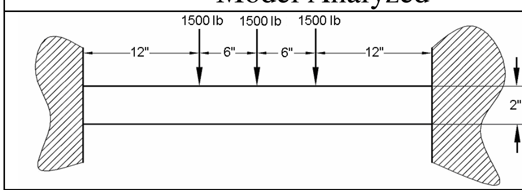
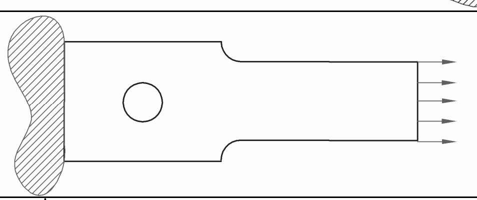
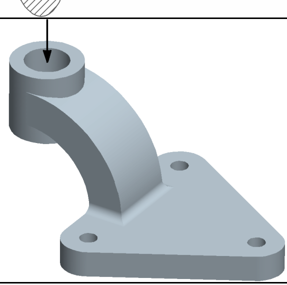
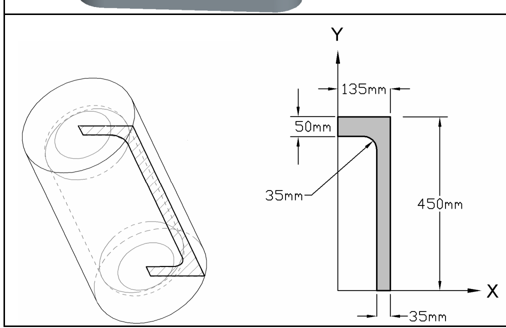
The prerequisites of the course are Applied Strength of Materials and Machine Elements. Therefore, the majority of topics covered in the FEA course are static structural analysis oriented. For example, the first FEA model that is discussed is the calculation of the deflection of a cantilevered beam. However, in order to correctly compare the FEA results with a manually calculated solution, the addition deflection topic of shear deflection[3] (which is not normally discussed in the Strength of Materials course) is introduced here. This provides the additional benefits of both reinforcing and expanding on a previously learned topic.



$$\delta = \frac{PL^3}{3EI} + \frac{6PL}{5AG}$$

Figure 2 Cantilevered beam is the first FEA model presented in the course, and its deflection equation that contains the additional term to account for deflection due to shear

During the semester seven different projects are assigned that require the student to analyze a part and write a formal report. Most of these projects require the student to compare their FEA result with a manual calculation or to do comparative evaluation between two FEA models. The following table illustrates several of the projects that have been assigned during the past year.

Model Analyzed	Requirements
	Determine deflection and stress, and compare to manually calculated values
	Use of symmetry boundary conditions and convergence, and compare to manually calculated values
	Determine maximum von Mises stress and deflection, using manual element sizing and convergence – also, evaluate stress in the fillet area as the radius → zero
	Use of axisymmetric elements, manually specify element size, and compare results with thick walled pressure vessel equations.

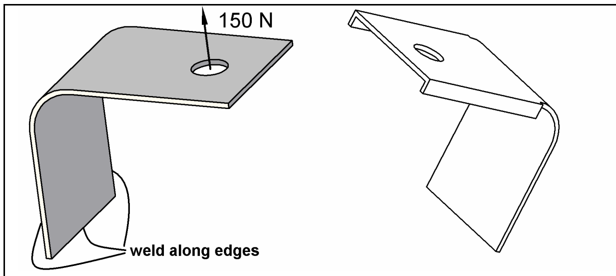
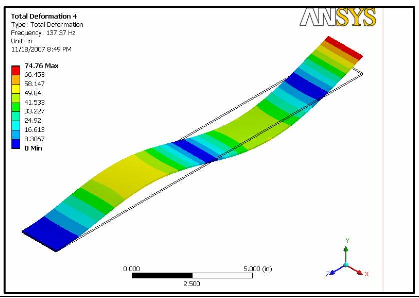
	Use of shell elements, in order to do a comparative deflection analysis between the two designs.
	Use of modal analysis to compare with manually calculated natural frequency values.

Table 1 Typical projects from the course used to illustrate different FEA concepts

Development of Instructional Material

There are many good finite element analysis textbooks[4,5,6] that are suitable for courses that are theoretically based. However, there were no books available that emphasized finite element modeling techniques and also included instruction on how to use ANSYS Workbench. Therefore, during the first semester of using Workbench, the instructional material consisted of a set of course notes that contained the finite element modeling concepts and a tutorial book[7] that illustrated the operation of the ANSYS Workbench program.

After this first semester, the decision was made to expand the course notes to include both the finite element modeling information as well as step by step instructions on using ANSYS Workbench. Because the author was working with an engineer who was responsible for the in-house training of ANSYS Workbench at a local corporation, the two decided to collaborate on the development of a textbook. This collaborative effort had the added advantage of bringing in a “real world” feel to the book. The result was a textbook[8] that is now used in both academic settings as well as for industrial training activities.

Because of the large number of steps required to define finite element models in Workbench, the tutorial portion of the book was written in a step-by-step format. This seems to work well, and serves as a valuable reference that students can use when constructing their own models. An example of the format of one of the steps is shown below.

Step 14 – Refine the mesh.

A. Duplicate the model. This is step is necessary to preserve the current mesh and results for comparison
B. Under **Model 2** in the Outline window, click the Right Mouse Button on **Mesh,** then select **Insert** and **Refinement**
C. Select the short edge at the base of the fillet as shown below
D. In the **Details** window pane change the **Refinement** value to ***2***
E. Click **Apply**

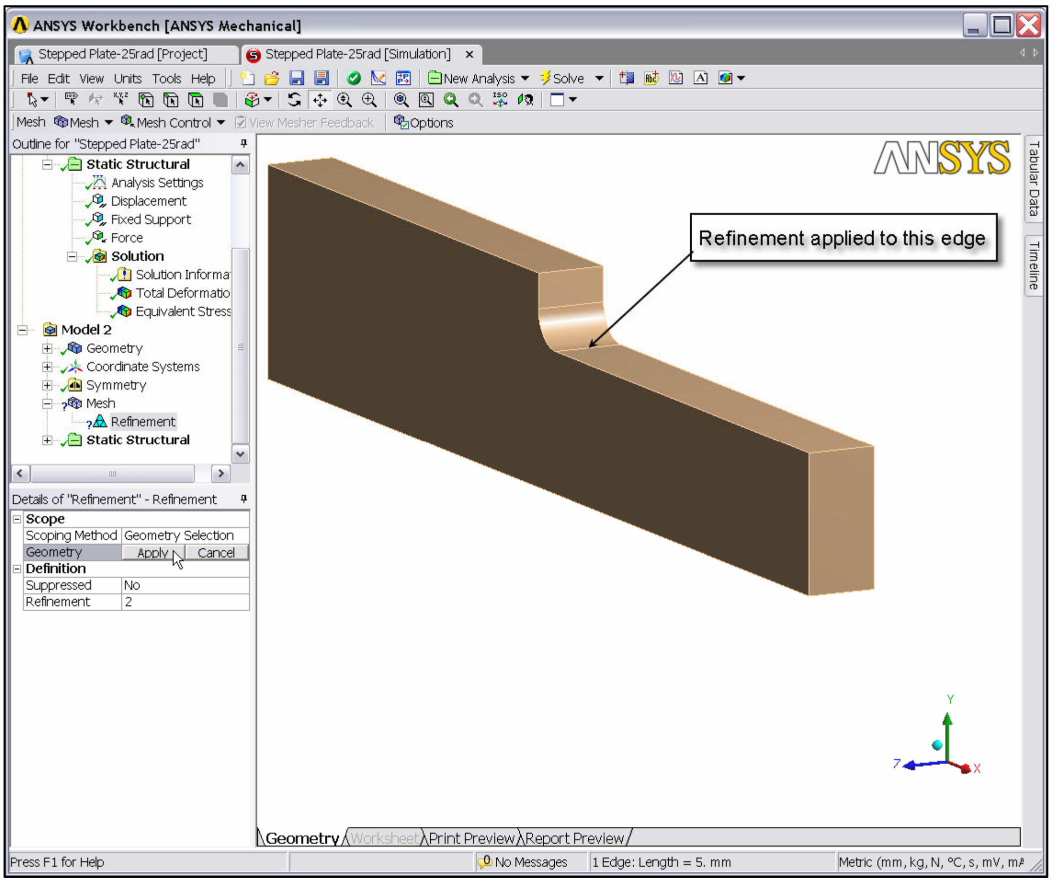


Figure 3 An example the step-by-step process contained in the tutorial

In addition to the traditional printed format, it was decided to also include a multimedia component with the book. This multimedia piece is in the form of "audio/video" files that illustrate the "real time" use of the Workbench program. It has been shown that when students have the experience of both seeing graphic images and hearing an explanation, their comprehension of the material increases dramatically compared to only reading text. In addition, the multimedia capabilities available with a CD-ROM based product allow many of the limitations that exist in conventional textbooks to be overcome. The audio/video files are designed to run independent of the ANSYS Workbench program, thereby allowing students to study at their own pace and to investigate and review topics of their own choosing.

Figure 4 below shows the retention benefit of media that contains both visual and audio stimuli, compared to media that contains only visual material.

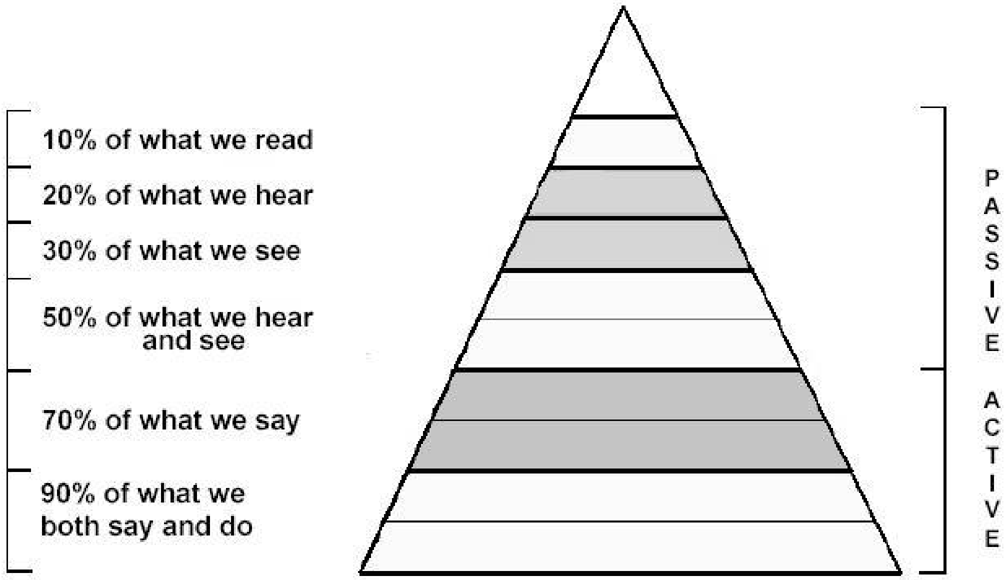


Figure 4 What we remember (from Edgar Dale's Cone of Learning[9])

It is expected that the multimedia format of the tutorial will have a positive impact on student learning based on the fact that the majority of engineering technology students are visual learners. Data reported by Felder and Brent[10] show that 85% of engineering students are considered to be visual learners. Although data could not be found for engineering technology students, the author expects that the percent should be even higher, due to the additional coursework in the areas of CAD and the emphasis on "hands on" laboratory work that is required in an engineering technology curriculum.

Development of these "audio/video" files begins by writing a script that is then recorded in a sound booth and converted into a wav file. The wav files are then edited and broken into "sound bites" so that they can be easily moved back and forth along a time line to correspond to the display of the visual image. The next task is to capture screen images that illustrate the sequence of steps that a user performs when running the ANSYS Workbench program. Finally, the series of bitmap images are combined with the audio "sound bites" using the Adobe Premiere program, which constructs the *avi* files used in the final product..

Conclusion

There is a lack of material that integrates both basic finite element modeling concepts with a tutorial approach to learning how to use a finite element program. The tutorial discussed in this paper is aimed at filling this void. It contains both written and multimedia material in order to accelerate the learning process. The examples contained in the tutorial make use of stress analysis problems seen in previous courses in order to give students "theoretical benchmarks" against which they can compare their FEA results. This allows the user to observe how changes to their models (such as varying the mesh size) will affect their 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, Englewood Cliffs, NJ, 1976

[4] Reddy, J. N., An Introduction to the Finite Element Method, 3rd Edition, McGraw-Hill, New York, NY, 2006

[5] Moaveni, S., Finite Element Analysis Theory and Application with ANSYS, 3rd Edition, Prentice-Hall, Upper Saddle River, NJ, 2008

[6] Hutton, D. V., Fundamentals of Finite Element Analysis, McGraw-Hill, New York, NY, 2004

[7] Lawrence, K.L., ANSYS Workbench Tutorial, SDC Publications, 2006

[8] Dadkhah, F. and Zecher, J., ANSYS Workbench Software Tutorial with Multimedia CD, Schroff Development Corp., 2007

[9] Dale, Edgar, *Audio-Visual Methods in Teaching*, 3rd Edition, Holt, Rinehart, and Winston (1969)

[10] Felder, Richard M, and Brent, Rebecca, "Understanding Student Differences", Journal of Engineering Education, Jan 2005