Combining Computer Analysis and Physical Testing in a Finite Element Analysis Course

William E. Howard, Thomas J. Labus, and Vincent C. Prantil
Milwaukee School of Engineering

Abstract

Finite element analysis (FEA) has become an essential tool in the product design process of many companies. A course in FEA is required in a large number of mechanical engineering and mechanical engineering technology curricula. Most FEA courses necessarily include some balance of theory and practical use of a commercial FEA program. In a course recently developed at Milwaukee School of Engineering, another element has been added to the FEA class in the Mechanical Engineering Technology Program: a mechanics of materials laboratory in which physical experiments are conducted to support the analysis exercises.

In this paper, the course content will be discussed, with emphasis on the specific lab exercises that allow measured results to be compared to FEA results.

Introduction

Finite element analysis is a subject area that is now commonly taught in Mechanical Engineering Technology (MET) programs. Several authors\textsuperscript{1,2,3} have reported on the development of undergraduate courses in mechanical engineering and mechanical engineering technology programs. Others\textsuperscript{4,5} have reported on efforts to add finite element analysis to traditional mechanics of materials courses. Most courses try to balance some amount of finite element theory with practice using a commercially-available software package. In this paper, the authors describe a course that adds a third component to a finite element course: a physical laboratory in which mechanics of materials experiments are performed and the results compared to FEA results where practical.

When the MET curriculum was revised recently at MSOE, the addition of the physical lab to the FEA course seemed to be a good fit. Although some of the reasons for considering this addition were logistical ones (elimination of one-credit stand-alone labs to make evening scheduling easier was a goal), the idea had merit for other reasons. For both the mechanics lab and the FEA course, one of the focuses has been the comparison of results to theoretical solutions. By integrating physical testing and FEA, additional comparisons can be made, and more complex problems can be considered. While performing FEA or mechanical tests for which closed-form solutions exist is obviously a necessary starting point, one of the main purposes of both FEA and
mechanical testing is to evaluate structural responses for which closed-form solutions do not exist. Analysis of more complex structures could be incorporated into the class if experimental results were available for comparison purposes. The new class, MT-3601, was offered for the first time during the Spring 2003 quarter.

Course Description

Most MET programs are considered to be more applications-oriented than mechanical engineering programs, with less emphasis on theory. It is possible to design a finite element analysis course with no theoretical content at all. Finite element analysis programs have become very easy to use. From many solid modeling programs, a few mouse clicks are all that is required to produce a stress analysis. Many software vendors advertise that their products are so easy to use that a draftsman can be quickly trained to perform stress analysis. This ease-of-use, however, has resulted in greater possibilities for misuse. An understanding of basic FEA theory is needed in order to select the most appropriate element type, evaluate the adequacy of the element mesh, apply boundary conditions consistent with the physical system, and interpret results. A physical testing lab further reinforces these concepts by allowing students to compare FEA solutions to measured data from the lab.

When developing the FEA course for MSOE, the following outcomes were desired. At the completion of the course, students are expected to:

- Be capable of using computer software to solve a system of simultaneous equations.
- Be able to assemble system equations for a simple finite element model, apply loads and boundary conditions, solve for unknown quantities and interpret results. Element types that the student will be able to use are spring, truss, and beam elements.
- Be capable of analyzing linear stress and heat transfer problems with ALGOR finite element analysis software.
- Be able to model simple structures with SolidWorks solid modeling software, mesh the surface with the ALGOR add-in, and export the model to ALGOR.
- Be able to measure strains from strain gages, and manipulate and interpret strain results.
- Be able to document results from FE analyses and physical tests in a logical fashion.

The new course incorporating the physical testing lab was offered for the first time during the Spring 2003 quarter. No textbook was required, but a text by Logan was used as a reference. (The Logan text is used in the FEA course in the Mechanical Engineering program at MSOE). The course was structured in a 3-2-4 format (3 hours lecture, 2 hours lab, 4 credit hours) and met for 2-1/2 hours two evenings per week. During these time periods, the class had access to a classroom, a mechanics lab, and a computer lab, so the allocation of class time was flexible from week to week. A typical week consisted of about 2 hours of classroom time and 1-1/2 hours each in the mechanics and computer labs. Activities of the ten weeks can be divided into three segments each three weeks long, with a final week of review and consideration of advanced topics. Class activities for each segment are described below.
Class Activities – Segment 1

Lecture topics for the first segment included steps of the FEA process, a review of matrix mathematics, and spring elements. Assemblages of springs, such as the one shown in Figure 1, are excellent structures for students to learn the steps of the FEA method. The math is simple, and results can be verified by concepts learned in statics classes. Students learned how to assemble element equations into a series of global equations, introduce boundary conditions, and solve for unknown displacements. The displacements were then used to find the force in each spring.

The first week’s computer lab was spent on the analysis of a shaft with a square cross-section subjected to a torsion load. The objectives of this lab were to make the students familiar with the ALGOR program and to demonstrate the effects of mesh density on the FEA solution. Students worked through an analysis to obtain the maximum shear stress in the shaft, using four elements per side (see Figure 2). Students then re-worked the problem with different numbers of elements per side assigned to groups of students. The results were then shared with the other groups and compared to the textbook solution for this problem. A comparison of the results from the different mesh densities is shown in Figure 3.

A point of emphasis during discussion of these results was for students to remember that finite element analysis is an approximate method and that many factors such as mesh density, element type, and boundary conditions affect the accuracy of the solution.

In the mechanics lab during the first week, stress and strain transformations were reviewed. Strain gages were discussed, and students were instructed in the use of the strain measuring equipment of the lab. Equations for strain rosettes were discussed, and the students were given an assignment to write a simple routine (spreadsheet or MATLAB code) to convert data from a

![Figure 1 Spring Element Example Problem](image1)

![Figure 2 Shear Stress Distribution in a Square Shaft](image2)

![Figure 3 Convergence of Solution: Square Shaft Problem](image3)
strain rosette into strains and stresses in a specific coordinate system. This routine was used the following week and again later in the quarter to assist the student with data analysis.

During week two, the mechanics lab consisted of tensile tests of three different tensile specimens. The first, shown in Figure 4, was a solid aluminum bar with multiple strain gages, including a strain rosette aligned at an angle to the longitudinal axis. Axial gages were placed on this specimen so that bending effects can be negated by averaging strain readings. A transverse gage allowed the Poisson’s ratio to be determined. Readings from the rosette were compared to those of the gages in the axial and transverse directions. The second specimen was similar in geometry to the first, except for a hole in the center in the specimen. A strain gage mounted on the inner surface of the hole (see Figure 5) allowed the axial strain to be measured there. The third specimen was more complex, with a slot and unequal-depth notches cut into the bar. Strain gages were located at the notch and slot radii. Loading for each specimen was applied by a hydraulic cylinder operated by hand. A pressure gage allowed the loading to be monitored and held at the desired level while each gage was read and the results recorded. The third specimen mounted in the loading fixture is shown in Figure 6.

Figure 4  Tensile Test Specimen With Strain Gage Locations Shown

In the computer lab of the second week, students performed analyses of two of the specimens from the mechanics lab. The plate with the center hole is a problem that is worked in many introductory FEA classes. Since an elasticity solution is available, the FEA solution can be evaluated for accuracy. From a modeling perspective, this was an easy exercise to introduce the use of symmetry. Also, the choice of stresses to display was discussed. In particular, students were asked to find the stress 90 degrees away from the location of the maximum stress. When the von-Mises equivalent stresses or maximum principal stresses are displayed, the fact that the stress is compressive is not apparent. By displaying the minimum principal stress or the normal stress transverse to the axis of the specimen, the magnitude of the compressive stress could be seen. The asymmetric third specimen from the mechanics lab was also analyzed. To save some time, students were provided a 2D drawing file in .dxf format to import into ALGOR. (Of course, the ability to import CAD files is an important feature of most FEA programs, and demonstrating that capability was of some value.) The purpose of this exercise was an
illustration of the importance of boundary conditions. While the symmetry of the plate with a center hole allowed modeling with a variety of constraint and load application methods, the asymmetric specimen required the application of boundary conditions that accurately simulated the actual conditions of the physical test. In particular, the specimen had to be modeled so that the bending of the specimen would be unconstrained. Boundary conditions for the model are illustrated in Figure 7, and the displaced geometry of the model is shown in Figure 8. Results from this exercise were excellent, with most students obtaining stress values from the mechanical and FEA labs within 5% of each other.

![Figure 5 Tensile Specimen With Center Hole](image1)

![Figure 6 Tensile Test of Asymmetric Specimen](image2)

![Figure 7 FEA Model of Asymmetric Specimen](image3)

![Figure 8 Deflected Shape of Asymmetric Specimen](image4)
During the third week, the FEA and mechanics labs were again matched, with column buckling the topic. In the mechanics lab, rods of different diameters, lengths, and materials were loaded in compression with a simple screw vise, and the buckling load recorded with a scale. End adapters allowed simulation of pinned and fixed end conditions (see Figure 9). A flat aluminum bar was also loaded in a compression testing machine, and the load-deflection curve plotted. This last test was simulated in the computer lab. Students learned how to use ALGOR to calculate the buckling load and plot the deflected shape.

![Pinned End Adapter Fixed End Adapter](image)

**Figure 9** End Adapters for Buckling Tests

Figure 10 shows the displaced shape of a bar pinned on both ends. Students computed the buckling loads for fixed ends (simulating the grips of the test machine) and compared it to the load recorded during the buckling test. The comparison of test results to hand calculations for these tests is interesting in that results for tests with pinned ends agreed very well with calculations (usually less than 5% error), while the test results for fixed-end tests were not as good (typically up to 20% error). Students learned that the idealized fixed boundary condition may not accurately reflect the actual geometry. A loose fit between the rod and the adapter allowed some rotation of the rod end, while the assumption in the calculated results is that the rotation is exactly zero. This lesson was applied to FEA: most real structural joints are not perfectly “pinned” or “fixed,” and care must be exercised in applying boundary conditions and interpreting results.

2.2 Class Activities – Segment 2

During weeks 4 and 5, the lecture material covered the formulation of truss element equations. The truss element was an easy step up from the spring element. Transformation equations were developed, and the concept of multiple degrees of freedom at a node was introduced. The trusses shown in Figure 11 were used as example problems. The fact that one of the trusses was statically indeterminate reinforced an important concept: since the FEA method is displacement-based, statically indeterminate structures are treated in the same manner as statically determinate structures. The formulation of the truss equations was repeated using an energy method. Application of the principal of minimum potential energy was more straightforward for the truss element than for the simple beam element, so using it here was a good lead-in to the development of the beam equations that began in week 6.
The computer lab of week 4 focused on truss analysis, with the example problems illustrated in Figure 11 analyzed with ALGOR and results compared to those of the hand analyses of the lecture material. During weeks 5 and 6, students began working with SolidWorks to model a beam section. Although solid modeling is not a prerequisite for this class, many of the students were already comfortable using SolidWorks. However, the geometry of the beam was simple enough so that new users could model the beam following detailed instructions. The ALGOR add-in to SolidWorks allows a surface mesh of a solid object to be created and then exported to ALGOR for the addition of loads and boundary conditions and meshing of the interior of the solid.

In the mechanics lab, a combined stress experiment was performed during week 4. This particular experiment was not directly linked with either the lecture or computer lab material, but allowed students to consider the combined effects of bending, torsion, and transverse shear loadings. The experimental apparatus is illustrated in Figure 12. Gages 2, 3, and 4 form a rosette, as do gages 5, 6, and 7.

During week 5, the T-beam section that will be analyzed with ALGOR was tested. The beam section and loading are illustrated in Figures 13 and 14. The load was applied with a hydraulic cylinder apparatus. Strain gages mounted at several locations between the loading points (where the moment was constant and the transverse shear force was zero) were monitored during the test. Results were compared to simple beam theory, and this beam was to be analyzed with FEA in the coming weeks. The mechanics lab time of week 6 was used for a mid-term exam.

**Figure 11** Truss Example Problems

*Figure 12 Combined Stress Fixture*
Class Activities – Segment 3

Development of the simple beam element continued into week 7 of the lectures. This beam element introduced several important topics:

- Students saw that direct equilibrium does not work well for elements more complicated than a truss.
- The shape functions were important for the first time. Shape functions were introduced with the spring and truss elements, but the linear variation of displacements made them trivial in these cases. For the beam, the shape functions were important for integrating strain energy through the element and for predicting displacement, slope, moment and shear between the nodes.
- Boundary conditions of slope, in addition to translational displacement, were incorporated.

The simple beam illustrated in Figure 15 was used as an example problem. A statically indeterminate beam was chosen for the example, again to illustrate the point that the deflection-based solution works for statically determinate and indeterminate structures equally well.
In the computer lab during week 7 (and during the time normally allotted for the mechanics lab), the T-beam tested earlier during the mechanics lab was analyzed, using solid elements created from the SolidWorks model developed earlier. The distribution of axial stress at the mid-span of the beam is shown in Figure 16. (Most of the elements have been hidden for clarity.) The linear distribution of stress through the depth of the section was of course the expected result.

The beam was also analyzed using simple beam elements. Results of these analyses and the test data from the lab allowed for some interesting comparisons, as shown in Table 1.

### Table 1 Results of Beam Analyses

<table>
<thead>
<tr>
<th></th>
<th>Axial Stress, Bottom of Beam, psi</th>
<th>Axial Stress, Top of Beam, psi</th>
</tr>
</thead>
<tbody>
<tr>
<td>From measured strains in mechanics lab</td>
<td>1,596</td>
<td>-3,549</td>
</tr>
<tr>
<td>From simple beam theory (hand calculations)</td>
<td>1,633</td>
<td>-3,924</td>
</tr>
<tr>
<td>From ALGOR analysis with beam elements</td>
<td>1,633*</td>
<td>-3,924</td>
</tr>
<tr>
<td>From ALGOR analysis with solid elements</td>
<td>772</td>
<td>-3,146</td>
</tr>
</tbody>
</table>

* Ratioed from stress at bottom of beam – results are calculated only for worst stress based on value of S (I/c) input for the cross-section.

The hand calculations agreed exactly with the beam element results. This was expected, since the beam element is based on the same assumptions as the simple beam theory used for the hand calculations. These results also agreed fairly well with the experimental results (about 10% error at the bottom of the beam). The results from the solid-element analysis were far off from the other results. Students were asked to consider what the differences might be. Some possible reasons discussed included:

- The simple beam calculations were made with a cross-section that neglected the fillets between the web and flange. The solid-element model includes the fillets, resulting in a stiffer structure. Although the solid-element model could be analyzed with the fillets suppressed for comparison purposes, ALGOR has a moment of inertia calculator that allows the effect of the fillets to be determined. The effect of the fillets on the moment of inertia is to increase it by about ½ of 1%. Therefore, the error introduced by neglecting the fillets was insignificant.

- Experimental errors, including reading of the applied pressure, locations of the supports and load application points, inaccurate modulus of elasticity, and strain gage errors,
caused the measured strains to be inaccurate. If only the solid-element model were being compared to the experimental results, this might have been the students’ conclusion. However, the agreement of the simple beam calculations and beam element model results to the experimental results cast doubt on the accuracy of the solid-element model.

- There are not enough elements through the thickness in the solid-element model to allow for the bending stresses to be accurately calculated. While this is a possibility, closer examination of the maximum and minimum stresses predicted by the solid-element model shows that the neutral axis location (assuming a linear distribution of stress) is more than ½ inch away from the centroid of the cross-section. This result suggests that some other type of loading is being introduced into the beam. With this in mind, the boundary conditions are suspect.

The model was analyzed with boundary conditions that allow only rotation about the y-axis permitted on the edge of the supported end, as shown in Figure 17. This boundary condition seems to be a good representation of the physical constraint, as the beam rests on a support that extends across the width of the beam, as shown in Figure 18. Note that the portion of the beam that extends beyond the support is not included in the finite element model.

However, the boundary conditions restrict motions that are possible with the physical constraint. In particular, the flange of the beam does not remain perfectly flat. Since the axial strain varies with distance away from the neutral axis, the transverse strain due to the Poisson’s ratio also varies. This variation of transverse strain results in curvature of the flange. Students can easily visualize this effect by bending a rubber eraser between thumb and forefinger and noticing the curvature transverse to the applied bending. To allow the model to curve in the transverse direction, boundary conditions were applied to the two corner nodes, as shown in Figure 19. (A boundary condition applied to a single node in the center of the flange would accomplish the same constraint, but rigid body rotations would be introduced.) The deflected shape of a slice of the beam section with these boundary conditions applied is shown in Figure 20. Although the deflections are greatly exaggerated, the tendency of the beam flange to curve rather than sit flat on the support is clearly evident.

*Proceedings of the 2004 American Society for Engineering Education Annual Conference & Exposition*

*Copyright © 2004, American Society for Engineering Education*
Stresses for the new analysis were 1,363 psi at the bottom and -3,429 psi at the top of the beam. These results are much closer to the experimental results than those of the previous analysis with solid elements.

An important lesson for the students to take away from this exercise was that solid elements are not always the best choice for an analysis. Many students think that because they have a part or assembly modeled with a 3D solid modeling program, it is logical to analyze the structure with solid elements. In this example problem, an analysis with over 14,000 solid elements produced no better results than an analysis with four simple beam elements. A frame made up of thin tubing sections, such as is typical for Mini-Baja and Formula cars built for Society of Automotive Engineers (SAE) student competitions, can require hundreds of thousands of elements to model with solid elements, when 20 or 30 beam elements will suffice.

During week 8, the use of finite element analysis to solve heat transfer problems was examined. A course in heat transfer is not a prerequisite for the finite element class, so the lecture material did not delve into the theory of the element development. Rather, a few basic concepts were introduced and analogies to structural analysis were made:

- The nodal temperatures are analogous to displacements,
- Heat flux is analogous to stress,
- Heat generation and convection result in nodal “forces”, and
- Specified nodal temperatures are the boundary conditions.

Students worked through a simple 4-element model with temperatures specified at two nodes and convection on one side. This example illustrated that the assembly and solution steps were similar to those of structural problems.
In the computer lab, a steel plate with a heat source and convection on one side was modeled with ALGOR. The model is illustrated in Figure 21. Results of the analysis, with nodal temperature distributions and heat flux, are shown in Figure 22. Students then worked through several variations of this problem. For example, the convection coefficient of the free side was changed from one corresponding to free air to one corresponding to forced air. In another version, a chain of elements from the heat source to the free side were changed from steel to copper. Students did not need to have studied heat transfer for the results of the analyses (lower temperatures for forced air, greater heat flux through the copper) to make sense.

In the mechanics lab during week 8, an experiment measuring beam deflections under a variety of loadings was conducted. These experiments further reinforced the earlier observations that simple beam theory produces accurate results for most beam geometries.

During week 9, a thick-wall pressure vessel was analyzed and tested. In the lecture portion, the advantages of using axisymmetric elements where allowed by geometry was discussed. Also, thick-wall pressure vessel equations were introduced, along with a numerical exercise to determine whether a vessel was “thick” or “thin” according to the theories. In the computer lab, the hydraulic cylinder from the mechanics lab was analyzed with axisymmetric elements. Results shown in Figure 23 include a displaced-geometry plot and the hoop stress distribution in a slice of the cylinder segment. In the mechanics lab, strains recorded from a strain gage rosette mounted on the outer surface of the hydraulic cylinder were recorded and analyzed.

Figure 21 Heat Transfer Example Problem

Figure 22 Results of Heat Transfer Analysis
Results for the hoop stress were very good (usually within 5-10% of the stress predicted by FEA and thick-wall theory) but axial stress results were not as good (typically 10-20% error). A discussion with the students reached the conclusion that the axial stress results were probably influenced by bending of the cylinder. Since the piston is simply run to the top of the cylinder, any flatness variation in the piston or cylinder caps would result in uneven loading of the cylinder wall. This result can be avoided by inserting a specimen into the fixture and loading it in tension so that the piston never reaches the end of the cylinder, but then any misalignment of the specimen can result in bending loads greater than those seen in the experiment. The best fix for this problem is to mount multiple axial gages around the circumference of the cylinder so that bending effects can be averaged out.

Class Activities – Final Week

In the last week of the quarter, advanced topics in FEA were introduced. Since all of the problems worked in this introductory course were linear, a discussion of the types of non-linearity encountered and how FEA is used to solve non-linear problems was presented. Also, the differences between p- and h-elements were discussed. Demonstrations of dynamic analyses using explicit-method codes were shown. The goal of this last week was to make the students aware of the capabilities of finite element analysis to solve a wide variety of the problems they are likely to encounter on the job, and to kindle an interest in the students to continue learning about FEA.

STUDENT FEEDBACK

Only five students were enrolled in the initial offering of this course during Spring 2003. (Most students enrolling for FEA had already completed the one-credit mechanics lab and so enrolled in the “old” course. Enrollment in future courses will be greater as students on the new curriculum track progress through the program.) These five students were asked to complete a standard MET Program assessment survey form. Results are summarized in the Table 2. Students were also asked if their interest in the subject was sustained or enhanced by taking the course. All five students answered “Yes” to this question. Written comments, although few in number, mentioned the applicability of the material to job requirements, and two students noted that the course refreshed/reinforced strength of materials topics.
<table>
<thead>
<tr>
<th></th>
<th>Strongly Agree</th>
<th>Agree</th>
<th>Somewhat Agree</th>
<th>Neither Agree nor Disagree</th>
<th>Somewhat Disagree</th>
<th>Disagree</th>
<th>Strongly Disagree</th>
</tr>
</thead>
<tbody>
<tr>
<td>I feel that the course taken previously as prerequisites prepared me well for this class.</td>
<td>1</td>
<td>2</td>
<td>1</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>I believe that the course content was consistent with the number of credit hours.</td>
<td>2</td>
<td>2</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>I can see the relationship of this course to others in the MET curriculum.</td>
<td>4</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>I believe that this course contributed to me career objectives.</td>
<td>4</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

In the lecture portion of the class, partial class notes were handed out. During the lecture, these class notes were filled in. This method freed the students from much of the copying of class notes from the board, and thus allowed the instructor to cover more material during a short lecture period. The authors have found that this method is especially well-suited to an FEA class, where the presentation of example problems involves many tedious and repetitious calculations. By having most of notes, students don’t waste time simply copying. However, having the students fill in some of the material keeps them actively participating. The students liked this method of note taking very much. There was no textbook required for the class, so having well-structured class notes was especially important.

RECOMMENDATIONS

Although the students were generally satisfied that the prerequisites of mechanics of materials and integral calculus were sufficient (the one student who disagreed was not specific), the instructor noted that the students were generally weak overall in computer skills, and poor with matrix math. As a result, students will be required to have a laptop computer in the future and more time will be devoted to matrix math at the beginning of the quarter. (Note: MSOE has a laptop computer program under which full-time students are required to lease a computer year-round and part-time students are required to rent a computer when enrolled in a class for which a computer is required.)

Some students seemed to fall behind more than in most classes. Most weeks had two lab sessions, and some students had a hard time keeping up, even though the workload for this 4-credit class was definitely less than for the 3-credit FEA course and 1-credit mechanics lab that it replaced. A more structured class schedule has been proposed for the next offering of the course, with some of the introductory ALGOR and SolidWorks material added to the “classroom” time and laboratory sessions limited to one per week, even though a couple of lab sessions will...
combine computer and physical labs. It is hoped that this more structured approach will help students to stay better organized.

The classroom topics to be covered are (a “class” here refers to one 50-minute block of time):

1. Introduction to finite element analysis (1 class)
2. Review of matrix math; computer solution of simultaneous equations (2 classes)
3. Introduction to ALGOR FEA program (4 classes)
4. Spring element formulation (2 classes)
5. Assembly and solution of equations (2 classes)
6. Failure criteria (1 class)
7. Effect of boundary conditions (1 class)
8. Truss elements (4 classes)
9. Introduction to SolidWorks (2 classes)
10. SolidWorks/ALGOR interface (1 class)
11. Beam elements (4 classes)
12. Heat transfer analysis (3 classes)
13. Introduction to advanced topics in FEA (1 class)
14. Examinations (2 classes)

The laboratory topics are:

1. Stress concentrations (Computer lab)
2. Strain gages, stress concentrations (Mechanics lab)
3. Column buckling (Computer/Mechanics lab)
4. Truss analysis (Computer lab)
5. Combined stresses (Mechanics lab)
6. Beam stresses with solid elements (Computer lab)
7. Beam stresses and deflections with beam elements (Computer lab)
8. Beam stresses and deflections (Mechanics lab)
9. Thick-wall pressure vessel (Computer/Mechanics lab)
10. Heat transfer (Computer lab)

Conclusions

The addition of a physical lab to a finite element course was implemented, with promising results. By providing experimental results for comparison, students can more clearly understand both the utility and limitations of finite element analysis.
Bibliography:


WILLIAM E. HOWARD
Ed Howard is an Associate Professor in the Mechanical Engineering Department and Program Director of the Mechanical Engineering Technology Program at Milwaukee School of Engineering. He holds a B.S. in Civil Engineering and an M.S. in Engineering Mechanics from Virginia Tech, and a PhD in Mechanical Engineering from Marquette University. He has 14 years of industrial experience, mostly in the design and analysis of composite structures.

THOMAS J. LABUS
Tom Labus is a Professor in the Mechanical Engineering Department at the Milwaukee School of Engineering. He holds a BS in Aeronautical Engineering from Purdue University and an MS in Theoretical and Applied Mechanics from the University of Illinois at Champagne. He has 35 years of industrial and consulting experience in the areas of mechanical design, fluid mechanics, hydraulics, high pressure engineering, and control of electro-mechanical and electro-hydraulic systems.

VINCENT C. PRANTIL
Dr. Prantil received his BS, MS, and PhD in Mechanical Engineering from Cornell University. His research interests lie in microstructural material modeling, finite element and numerical analysis. He was a senior staff member at Sandia National Laboratories California in the Applied Mechanics and Materials Modeling departments for ten years. He joined the mechanical engineering faculty at MSOE in September 2000.