A Project-Based Introduction to the Finite Element Method

S.M. Miner, R.E. Link
United States Naval Academy

Abstract

All mechanical engineering majors at the Naval Academy are required to take a course in Computer Aided Design during their senior year. The underlying philosophy of the course is to introduce students to computer based solution techniques that are currently used in engineering practice. To emphasize the utility of the computer the problems selected for solution are ones that would be difficult to solve by hand. In particular, students are introduced to the finite element method through a project requiring the design of a bracket that must meet size, load, and deflection requirements. In addition, the strength to weight ratio of the bracket is to be optimized. The students are given a brief introduction to the fundamentals of the finite element method, including basic theory and practical guidelines for modeling. Then design teams are formed to design and build the brackets. The brackets are tested to check the performance against the design requirements, to compare the teams predicted results to actual performance, and to see which team achieves the highest strength to weight ratio. The IDEAS software package is used to generate the geometry of the bracket and to perform the finite element analysis. Design teams have developed brackets with strength to weight ratios of 6000 and have been able to predict failure loads to with 10% of the measured value. This project-based approach to the finite element method gives the students an appreciation for how powerful the method can be in performing structural analysis.

I. Introduction

The past twenty years has seen a rapid advancement in the capability of computer-aided design tools. Commercial software packages are readily available to assist with all phases of the design process from ideation through synthesis and analysis, detail design and testing to prototype and production. Computer-aided design tools have become an essential part of the modern design and manufacturing environment and engineering curricula has evolved to include instruction in this field. Virtually all engineering schools include instruction in computer-aided design to some extent. Design software has become so powerful that a novice can conduct sophisticated analyses without knowing very much about the details or limitations of the analysis process.

While it is important for engineering schools to educate students about the use of computer-aided design tools, they must also ensure that the students have a basic understanding of the underlying principles upon which these computer programs are based. Striking a balance between teaching the fundamentals and giving the students hands-on experience with the technology continues to be a challenge. The real benefit in introducing the students to the technology is the ability to solve more interesting, physically realistic problems in a short amount of time.

All Mechanical Engineering students at the Naval Academy are required to take a course in computer-aided design in the fall semester of their senior year. Roughly one quarter of the course is devoted to introducing the finite element method. Students use the finite element method to complete a project requiring the design of a bracket that must meet size, load, and deflection requirements. In addition, the strength to weight ratio of the bracket is to be optimized. The objective of this paper is to illustrate how a project-based introduction to the finite element method using a commercial software package, SDRC I-DEAS, provides a basic introduction to the theory, as well as, a meaningful experience using the technology.

It is important for the student to have a basic understanding of the finite element method, otherwise it can be difficult to evaluate the results that the commercial packages produce. By understanding the basic approach of the modeling, the student can make a more informed interpretation of the results to decide if they make sense. All too often, students and professionals alike, readily accept the results generated by the computer without casting a critical eye at them.

At the U. S. Naval Academy the Computer-Aided Design course is heavily oriented towards team design projects. The lectures are organized to develop the new material the students will need to learn, in a logical manner that parallels their use of the software in the laboratory portion of the course. Ample time is provided once the new material has been presented to allow the students to apply it to their projects. In addition, a few short lab exercises and a homework assignments are given to reinforce the concepts developed in the lectures.

II. Basic Theory

As an introduction to the finite element method the students are given a set of notes[1] (http://web.usna.navy.mil/~link/fea.pdf, http://web.usna.navy.mil/~link/fea2.pdf) that are covered during two lectures. The following six major steps in the finite element process are emphasized:

Step 1. Establish Governing Equations and Boundary Conditions.
In order to generate a valid approximate solution to a problem, the differential equation that governs the behavior and the corresponding boundary conditions for the problem must be determined. Once this is done the appropriate finite element formulation can be used to generate the solution.

Step 2. Divide Solution Domain into Elements.
In this step the entire solution domain is subdivided into “small” elements. Care is taken to make sure that enough elements are included to capture the behavior of the solution over the entire domain. Areas of particular interest and care are locations where critical values are expected, locations with large gradients, locations where the geometry changes suddenly, locations where boundary conditions and loads are applied. Typically, the larger the number of elements the better the approximation of the solution to the differential equation.

Step 3. Determine Element Equations.
Once the elements are formed, the algebraic equations to be solved are developed for each individual element. The form of the algebraic equations for every element will be the same. Differences from one element to the next will be due to changes in element size and properties. This is the power of the finite element method, the equations can be written once for a general element then they only need to be modified to reflect a particular elements geometry and properties.

Step 4. Assemble Global Equations.
Once all the element equations are generated they are put together to form a system of equations for the entire solution domain.

Step 5. Solution of Global Equations.
This system of equations is solved for the value of the dependent variable in the original differential equation at discreet points throughout the solution domain. Depending on the problem type there may be hundreds, thousands, tens of thousands, or even hundreds of thousands of points at which the solution to the differential equation is approximated.

Step 6. Solution Verification.
The accuracy of the solution must be verified before the results can be considered valid. One way to do this is to refine the mesh (increase the number of elements) and rerun the solution. If the value of the dependent variable at the discreet points in the mesh does not change significantly as the mesh is refined, the solution is deemed to be accurate.

As a means of illustrating the finite element method, the six steps are related to the analysis of an axially loaded bar. Figure 1 shows the bar and the boundary conditions used and figure 2 shows the corresponding finite element representation. The desired results are the displacement and stress distribution along the length of the bar. This example is convenient because it allows all the steps of the modeling process to be demonstrated, derivation of element equations, calculation of element matrices, assembly of the global equations, contributions of multiple elements to a nodal equation, solution of the global equations, and comparison to the exact solution. After covering this material in lecture the students are given a similar problem for homework that reinforces the concepts from the lecture. They are asked to determine the displacement, strain, stress, and force in an axisymmetric rod with a known displacement at the free end. They generate and solve by hand a three element model. They also derive the exact solution to the governing equation and compare the finite element results for displacement and stress to the exact solution.

While they are covering the basic theory of the finite element method in the lecture, the students are getting hands-on exposure to finite element modeling software in the laboratory. During two laboratory periods the students complete workshops 11A through 11C in the IDEAS Student Guide[2], which introduces them to meshing, boundary conditions, solutions and post processing.

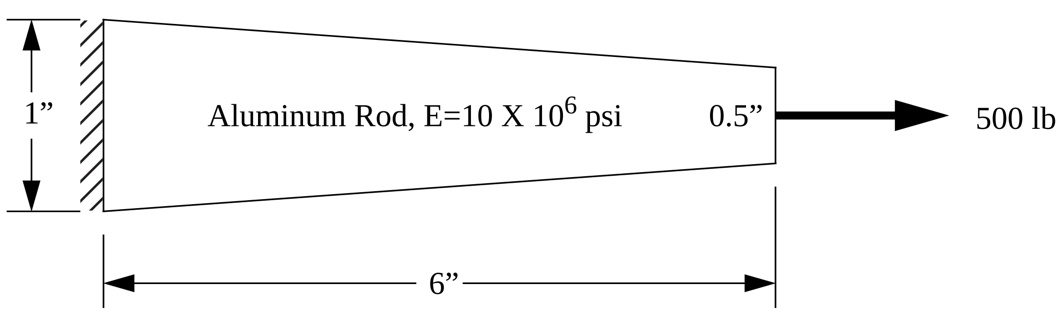
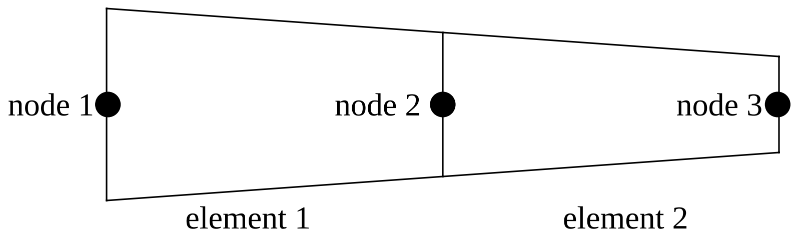
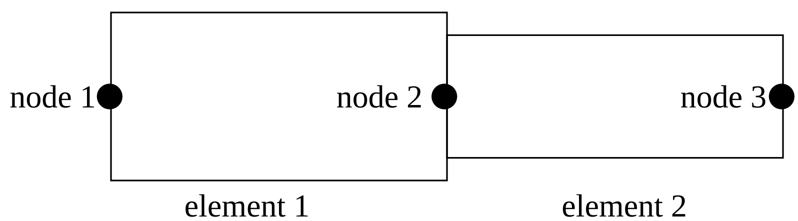


Figure 1 Axially Loaded Rod



Element Subdivision



Constant Property Approximation

Figure 2 Finite Element Division

III. Bracket Design Project

The culmination of the this brief introduction to the finite element method is the design and testing of an optimized support bracket. The project description is:

EM477 Computer-Aided Design, Fall 1998
Finite Element Analysis Design Project

Introduction: A top secret program is underway to develop the next generation fighter aircraft. Weight reduction of all structural components is a key issue in this design. Every ounce of material should be used to its fullest extent to maximize the payload carrying capacity of the aircraft. The designers have spared no expense in achieving this goal. They have even developed a new material, transparent aluminum, for special structural applications. One component made out of this material is the Auxiliary Attachment Bracket shown in Figure 3. This bracket connects a payload to the underside of one of the wings of the aircraft. It is attached to the wing by two 0.375 in. dia. steel pins spaced 7 inches apart as shown. The payload is attached by another 0.375 in. dia. steel pin. When the bracket is mounted in position, it restricts access to an inspection cover located on the fuselage just behind the bracket. The aircraft mechanics must be able to access this cover even when the bracket is in place, so a hole was cut into the bracket to permit access. Early prototypes have been experiencing repeated failures of the bracket because of the high stress concentration at the sharp corners of the opening. You have been tasked to redesign this bracket to optimize its strength-to-weight ratio. The bracket must be able to support a force, F, of 500 lbs directed as shown. The deformation of the bracket must be limited so that the pin at C does not displace more than $\delta = 0.200$ in. when the load is applied or the payload will come into contact with other critical components.

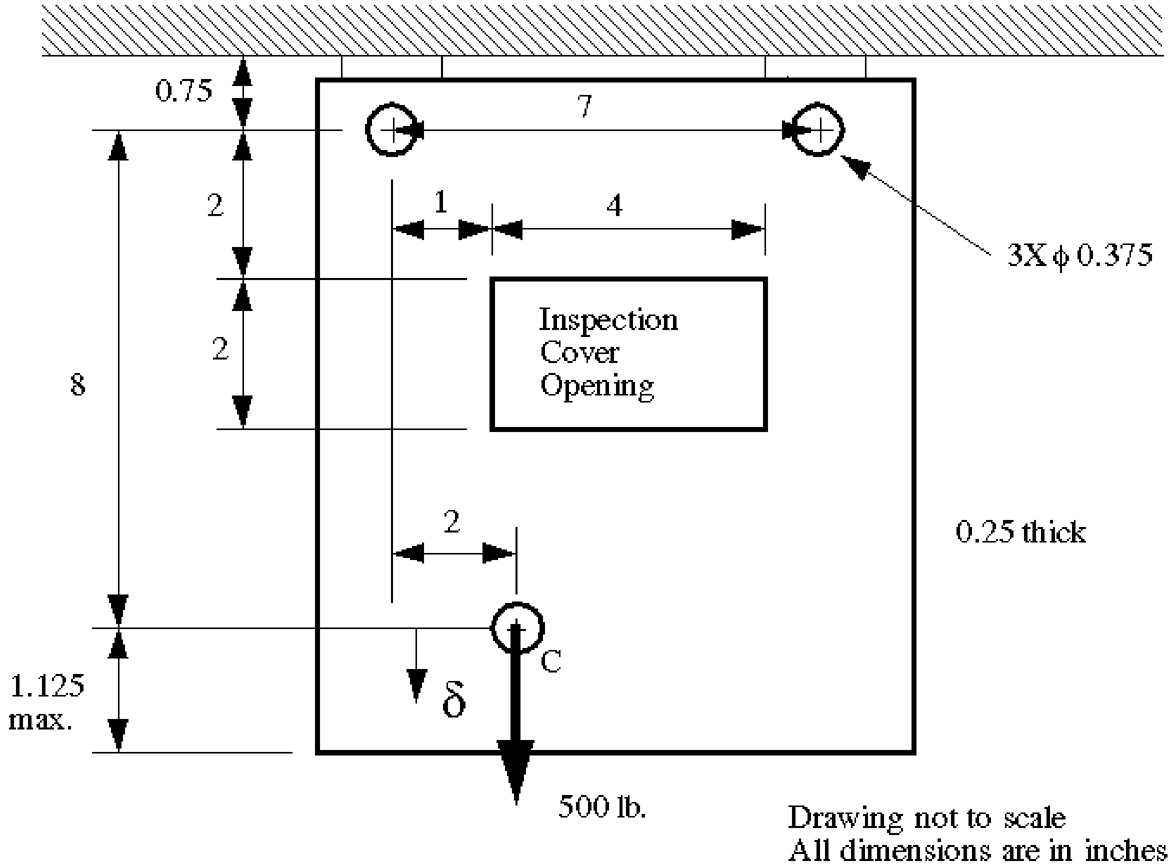


Figure 3 General dimensions for Auxiliary Attachment Bracket

IV. Results

The geometry and finite element mesh for a typical design that has been optimized is shown in Figure 4. Eight-node quadrilateral, thin shell elements were used for the bracket. The boundary conditions were modeled by restraining the displacement of several of the nodes around the top portion of each of the upper holes and by distributing 500 lbs. of force among several of the nodes located on the lower edge of the bottom hole. The model was solved and the deflection of the load point and the stress distribution were investigated. The maximum deflection of the lower pin for this model was 0.137 in., which was less than the 0.2 in. deflection that was permissible.

The distribution of the von Mises stress is plotted in Figure 5. In general, the stresses are well below the yield stress of 9 ksi for the bracket material. There are a few extremely localized regions where the von Mises stress exceeds the yield strength of the bracket. An example is shown in Figure 6 which is an enlargement of the area around the lower pin hole. These high stress regions would lead to local plastic deformation but would not in general lead to complete failure of the bracket. Bracket failure would occur when the net section of the ligament reached full yield. A rough estimate of the failure load could be made by scaling the results for the 500 lb. load to determine the load at which all points across the ligament exceed the yield strength. It is understood that a more accurate and proper analysis would include a non-linear analysis of the model to include yielding and plastic deformation, but that type of analysis is beyond the scope of this introductory project.

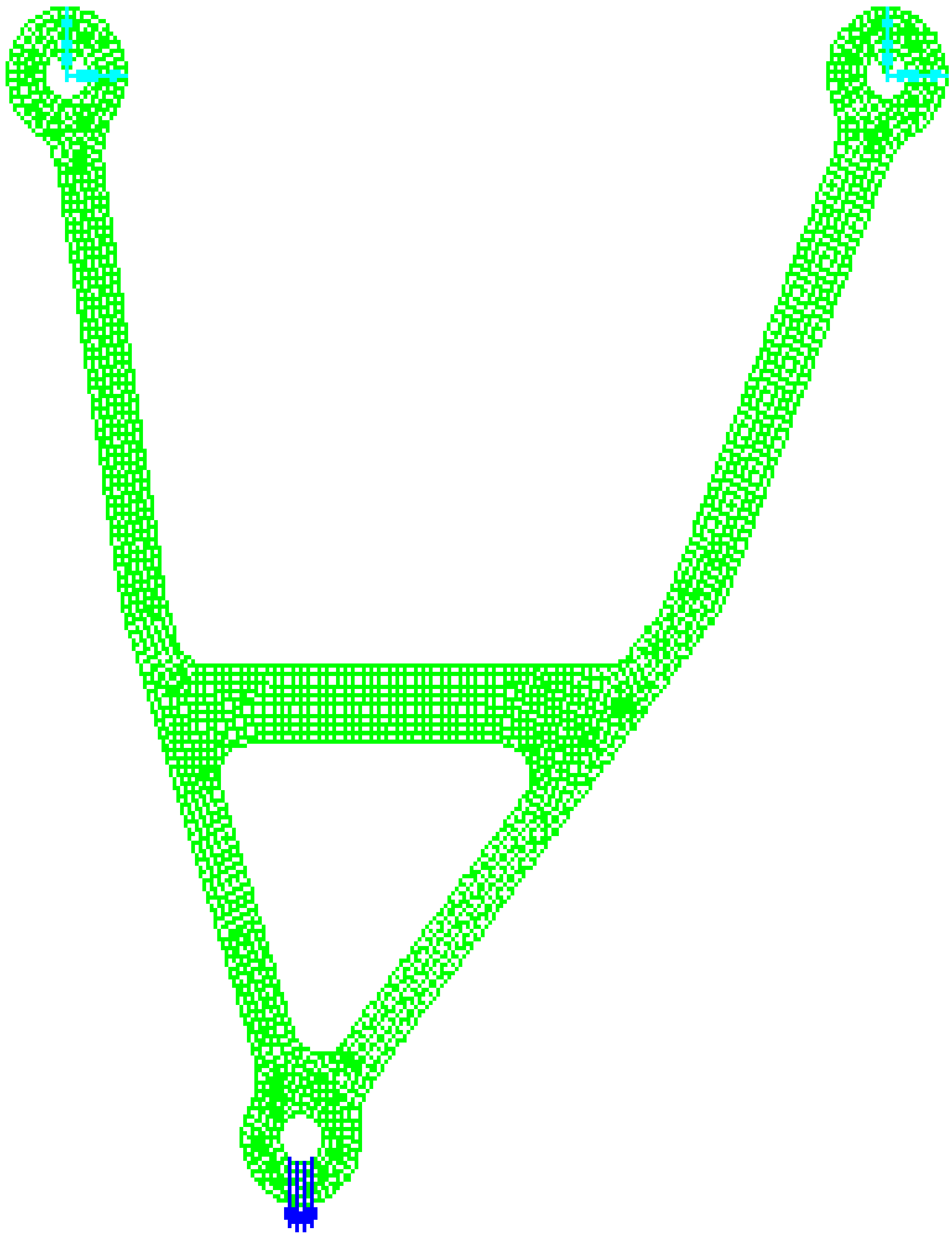


Figure 4 Bracket Mesh

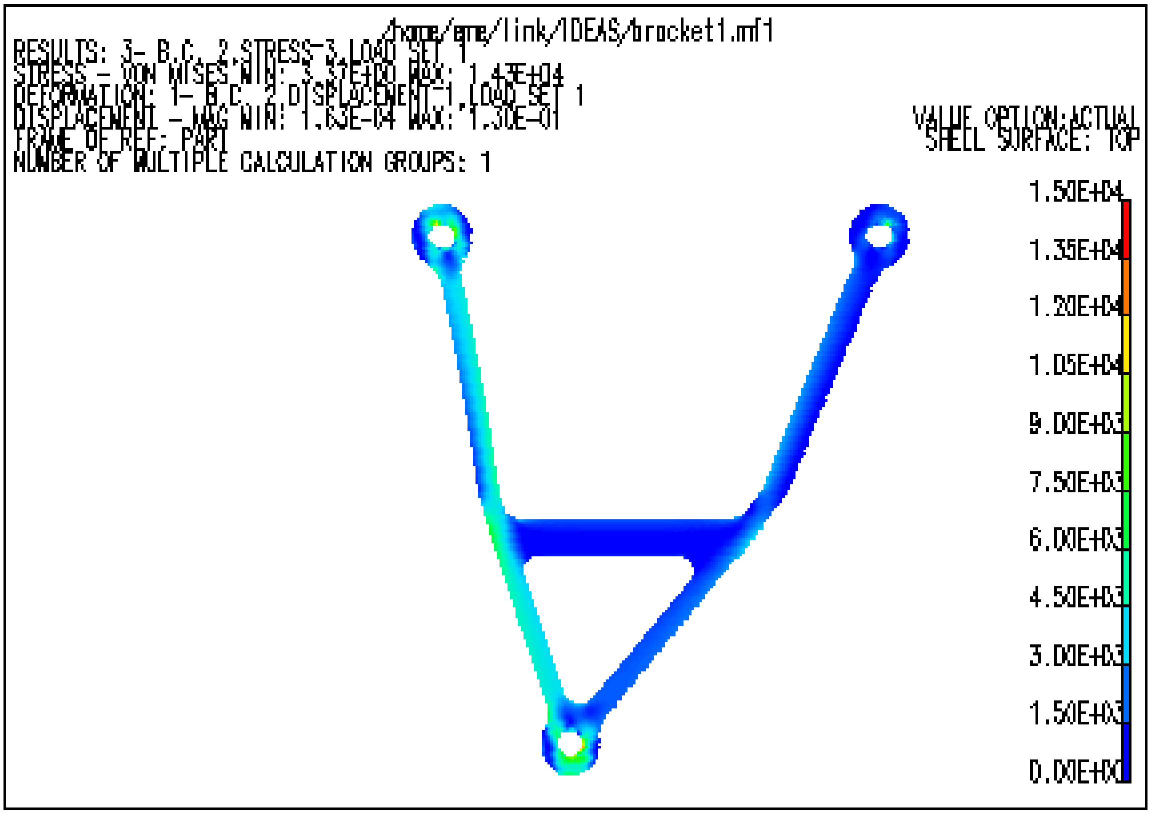


Figure 5 Bracket Stress Results

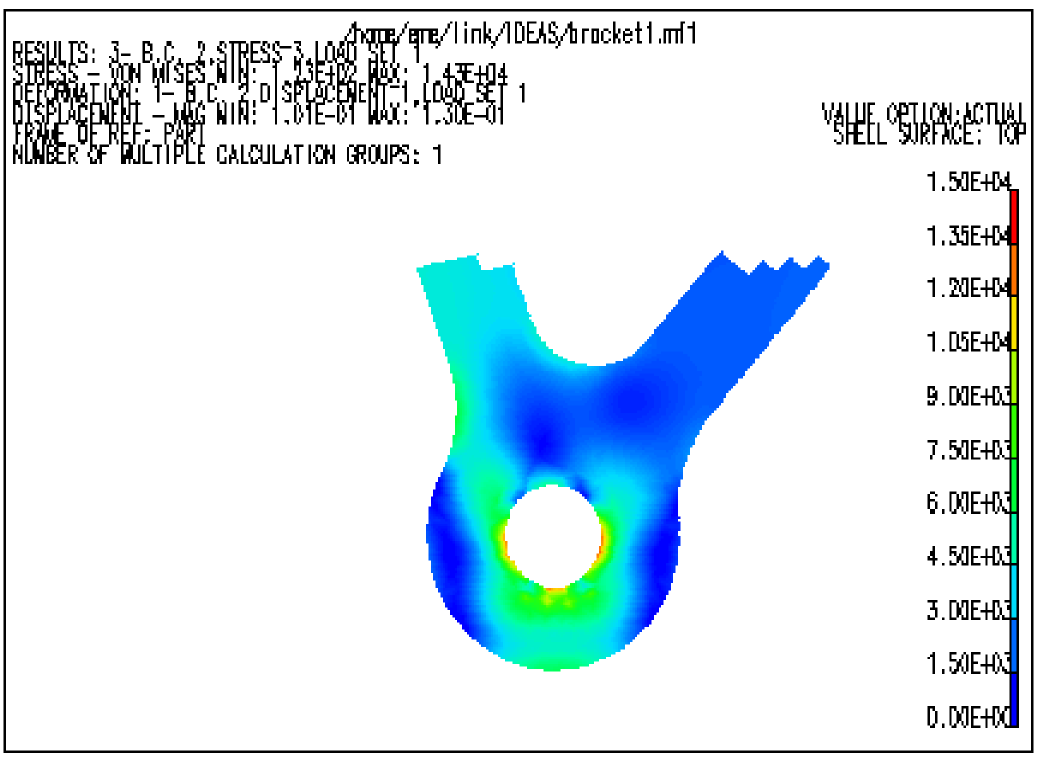


Figure 6 Close-up of Bracket Stress Results

Table 1 shows the results of the testing for all design groups from the fall 1998 offering of the course. Results show that the deflection at the 500 lb load was typically under predicted with half of the groups coming within 30% of the actual deflection. In the prediction of the failure load half of the groups were within 15% of the measured value. It should be pointed out that elastic analysis was used in all cases. No attempt was made to address plastic effects, which was beyond the scope of this introduction. However, it was still very instructive for the students to see the capability of the finite element method.

Team	Weight (lbs)	δ @ 500 lb Pred./Actual	Max. Load Pred./Actual	Strength/Weight
1	0.204	.060/.062	1500/1593	7808
2	0.206	.090/.123	1650/1560	7558
3	0.243	.078/.099	1700/1761	7242
4	0.237	.048/.062	1500/1632	6886
5	0.142	.093/.119	1125/1041	6271
6	0.187	.000/.062*	670/1167	6241
7	0.212	.000/.083*	1187/1311	6184
8	0.149	.180/.065	1192/804	5396
9	0.302	.000/.077*	510/1542	5106
10	0.270	.078/.108	1500/1368	5066
11	0.245	.105/.062	1200/1230	5020
12	0.258	.000/.055*	610/1233	4779
13	0.230	.200/.058	875/1020	4439
14	0.213	.060/.080	1500/933	4380
15	0.396	.038/.053	3000/1686	4252
16	0.149	.130/.077	830/609	4087
17	0.442	.160/.058	2375/1512	3421
18	0.336	.200/.080	2197/1128	3357

* Not all groups were asked to provide an estimate of deflection

Table 1: 1998 Bracket Design Results

V. Summary

All mechanical engineering majors at the Naval Academy are required to take a course in Computer Aided Design during their senior year. The underlying philosophy of the course is to introduce students to computer based solution techniques that are currently used in engineering practice. One part of this course which uses a bracket design project to introduce the students to the finite element method was described. Through a combination of lecture and laboratory experiences the students gain an appreciation for the basic theory and its application. In particular, they gain an understanding of issues related to meshing, the influence of boundary conditions, and the interpretation of results. This project-based approach to the finite element method gives the students an appreciation for how powerful the method can be in performing structural analysis.

Bibliography

1. Link, R.E. and Miner, S.M., EM477 Computer-Aided Design, Lecture Notes, 1999.
2. Lawry, M.H., I-DEAS Master Series, Student Guide, Structural Dynamics Research Corporation, 1998.

STEVEN MINER
Steve Miner is an Associate Professor of Mechanical Engineering at the United States Naval Academy. He received hi B.S., M.S., and Ph.D. in Mechanical Engineering from the University of Virginia. Before coming to the Naval Academy he worked as a Senior Engineer for Westinghouse Electric Corporation designing cooling systems for avionics. Currently, he teaches courses in the design sequence at the Naval Academy.

RICHARD LINK
Richard E. Link is an Assistant Professor of Mechanical Engineering and the Faculty Director of the Computer-Aided Design and Interactive Graphics laboratory at the United States Naval Academy. He received a Ph.D. in Mechanical Engineering from the University of maryland in 1993. Prior to joining the faculty at USNA in 1995, he spent ten years as a Senior Mechanical Engineer in the Fatigue and Fracture Branch at the U.S. Navy's David Taylor Research Center in Annapolis, MD. He teaches courses in mechanics, computer-aided design, and manufacturing.