Session 3568

Applications of Finite Element Analysis for Undergraduates

C.J. Lissenden, G.S. Wagle, and N.J. Salamon

Department of Engineering Science and Mechanics, Penn State University

Abstract

The Engineering Science and Mechanics department and the Mechanical Engineering department at Penn State share responsibility for teaching an undergraduate course on finite element analysis applications. We present one approach for teaching this course. Instructors can approach a course like this in a variety of ways. Faculty, students, and industry generally disagree as to what the learning objectives of this course should be. Furthermore, it is very difficult to get consensus from any one of these three groups. Should the focus be on using commercial software? On writing software? On the finite element method itself? On understanding finite element results? Our course objectives are to produce students capable of undertaking linear finite element modeling, who understand the basics of how commercial software packages work and the results they give, and what errors could be present. We describe the course content, which includes a mix between the finite element method and applications using a commercial software package. Special attention is given to each of the four projects that are assigned during a semester, with emphasis on learning objectives, project specifics, and student results. The students use the PRO/MECHANICA software package for these projects. While these projects change somewhat from semester to semester they generally cover: (1) plane stress elements, (2) axisymmetric elements, (3) frame elements, and (4) solid elements. In the most recent semester, students were provided a solid model of a bicycle crank arm for one project and were asked to perform a stress analysis of the crank arm. Students presented their results to students in an advanced mechanics of materials course who were designing a crank arm for their class project. In this way students were introduced to how finite element modeling fits into the design process.

Background

The course, "Applied Finite Element Analysis" is a technical elective for undergraduates at Penn State University. It is cross-listed as an Engineering Mechanics course and a Mechanical Engineering course. While different instructors treat the course differently, the first author requires only that students have taken an elementary mechanics of materials course. The first two thirds of the semester are devoted to traditional engineering education; in-class lectures, textbook reading, homework assignments, and in-class examinations. The focus of the class is on the finite element method itself, which is broken down into seven steps:

1. Discretize the continuous system into finite elements
2. Describe the element connectivity at the nodal points
3. Determine the element response in terms of nodal variables
4. Assemble the global system of equations

5. Apply the boundary conditions
6. Solve the system of equations for the primary variables
7. Post-process to determine the secondary variables

Particular emphasis is placed on step three. The principle of stationary potential energy is used to develop a library of elements for analysis of: bars, trusses, beams, frames, plane problems, axisymmetric problems, and 3-D solid problems.

The final one third of the semester focuses on applying FEA to mechanics and mechanical engineering problems and is all project-related work. Classes are held in a computer lab, where demonstration problems are worked and projects are assigned. The 3-D stress analysis project is linked to a design project for concurrent advanced mechanics of materials course[1].

Projects

The following pages contain Attachments 1-4 that give the statements of projects 1-4, respectively, assigned in the Fall semester of 2001. Also provided on these attachments are the learning objectives associated with each project.

Verbal comments from students to the instructor indicate that the students enjoy working on the projects, that the projects clarify parts of the lecture material, and that the projects are very valuable to them. Of course they get to learn how to use a commercial finite element package, which is what many of them expect when they register for the course.

Bibliography

1. C.J. Lissenden, G.S. Wagle, N.J. Salamon, 2002, "Design project for advanced mechanics of materials," 2002 ASEE Annual Conference Proceedings, Session 2468.

Biographical Information

CLIFF J. LISSENDEN, Ph.D. (University of Virginia, 1993) is an associate professor of Engineering Science and Mechanics at Penn State. In addition to teaching engineering mechanics courses ranging from statics to plasticity theory, he performs experimental and modeling studies of material response in the presence of multiaxial stress states. He is a member of ASEE, SES, ASME, ASCE, and Sigma Xi.

GAUTAM S. WAGLE is a Research Assistant at Penn State. He received a B.E. degree in Mechanical Engineering from the University of Bombay, India in 1997. He worked for a year as a Pre-planning engineer in the Switchgear manufacturing division of M/s Larsen and Toubro Ltd, Bombay, India. Gautam received his M.S. in Engineering Mechanics from Penn State in 2000.

N.J. SALAMON, Ph.D. (Northwestern University, USA) has been a professor at Penn State since 1985. Prior to that he was associate professor at West Virginia University and assistant professor at the University of Wisconsin Milwaukee. He has taught mechanics at the undergraduate and graduate level since 1975 and is a proponent of project work in engineering classes, in particular design. He does research in stress analysis of materials and structures with the emphasis on computer simulation. He is a fellow of the ASME and member of ASEE. His hobby is hiding away in forests.

Attachment 1 – Project 1: Plane Stress Analysis of Short Cantilever Beam

Here we will explore convergence issues by considering the cantilever beam of problem 5.9 in the textbook (p. 169). The aluminum (E=70 GPa, ν=0.33) beam is 60 mm long, 30 mm deep, and 10 mm wide. A transverse force of 10 kN is distributed along the free end of the beam.

1. Discretize the beam into two triangular elements as we did for homework. Set the convergence method to "Multi-Pass Adaptive", the "Polynomial Order Min." to 1, and "Polynomial Order Max." to 1. This will prevent the program from doing the convergence check and will simulate CST elements.
2. Refine this mesh into a uniform mesh having 32 triangular elements. Again analyze this model with 1^{st} order polynomial elements.
3. Now take the 2-element mesh from above and set the maximum polynomial order to 9 and re-analyze the model. Pro/M will automatically check for convergence. Set convergence to 5% on local displacement, local strain energy, and global RMS stress.
4. Take the 32-element mesh from above and set the maximum polynomial order to 9 and re-analyze the model. Pro/M will automatically check for convergence.

Plot the flexural stress (σ_x) and shear stress (τ_{xy}) contours on the deformed model for each of these four cases. Also plot the order of the interpolation polynomial (P-level) along the element edges for #3 and #4 above. Tabulate the flexural stress at point *A*, shear stress at point *B*, and deflection at point *C*. Compare with elementary beam theory. A table is provided on the cover sheet.

Objectives:

- Learn the importance of obtaining a converged solution with either h-elements or p-elements,
- Verify the software's solution with hand calculation,
- Show that elementary beam theory has limitations,
- Get students accustomed to the software on a problem having a simple geometry.

This problem had been assigned for homework earlier. Students discretized the beam into two CST elements and compared their FEA solution with elementary beam theory.

Attachment 2 – Project 2: End Plug Design

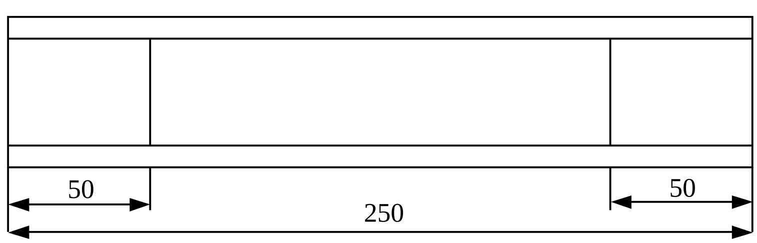
A thin-walled cylindrical tube (ID = 40mm, OD=45mm, L=250mm) fabricated from a fiber-reinforced aluminum composite material having the following properties is to be tension tested (to 50 kN) in your laboratory. The fibers are aligned in the hoop direction [90º].

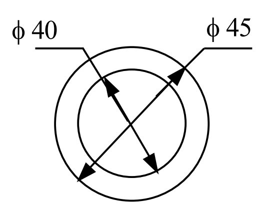
$E_{longitudinal}$ = 240 GPa
$E_{transverse}$ = 130 GPa
$\nu_{longitudinal}$ = 0.33
$\nu_{transverse}$ = 0.19
$G_{longitudinal}$ = 50 GPa
$G_{transverse}$ = 55 GPa

The tube ends are gripped by two collets (chucks) that are 50 mm long. The grip pressure applied by the collets is 30 MPa. End plugs are used to prevent excessive deformation and cracking in the tube due to the grip pressure. Your current end plugs are simple cylinders 50 mm long and made from steel (E = 200 GPA, ν = 0.3). **Try to improve the design of these end plugs by reducing the longitudinal stress (σ_z) in the tube.**

The results that you need to turn in are:

1. The finite element model (showing the elements, boundary conditions, and applied loads) used for the current design and the new design.
2. Contour plots of the longitudinal and hoop stress components (σ_z and σ_r) shown on the deformed mesh for both the current design and the new design.
3. Comparison of the maximum longitudinal and hoop stresses in the tube for the current design and the new design.
4. The order of the largest polynomial (3, 4, … 9) used in the analysis and where it occurred in the model.
5. A design sketch of the new end plug.



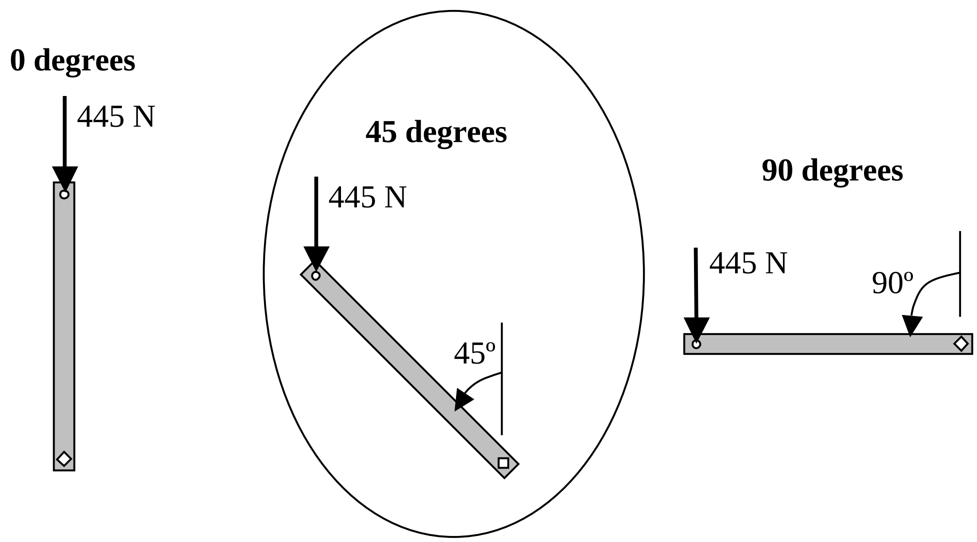


Objectives:

- Use FEA to improve an existing design,
- Become familiar with axisymmetric modeling,
- Become familiar with using composite materials.

Attachment 3 – Project 3: Crank Arm Analysis

EMch 400 "Advanced Strength of Materials and Design" students have designed a crank arm for a bicycle that is straight and has a circular cross-section. This crank arm has been manufactured from a Pro Engineer solid model (Fig. 1a), instrumented with two strain gage rosettes, and tested in the laboratory. A 100 lb (445 N) force was applied with the crank arm in five different positions and strains were recorded. The EMch 400 students need to compare their design with that of a real crank arm. A Pro Engineer solid model has been created of a real Shimano crank arm (Fig. 1b). The two solid models have been transformed into Pro Mechanica models that contain only the geometry.
These two files will be provided to you via email. You must discretize (using a maximum aspect ratio = 10), apply material properties, constraints, loads, and analyze the two crank arms in one of the three positions shown below. The specifications are: material – 7075-T6 aluminum (E = 72.4 GPa, $\nu = 0.33$), length of crank arm from centerline of crank shaft to centerline of pedal spindle – 175 mm, distance along pedal spindle from centerline of crank arm to applied force – 95 mm, force – 445 N applied in the direction of gravity, constraints – crank arm is assumed to be fixed to crank shaft. The units used in the geometry files are mm. Thus, if you use mm, N, and s; then the stress results will be in MPa. The diameter of the circular cross-section crank is 25 mm, while the real crank is curved and the cross-section varies. You need to only consider the crank orientation that is circled.



The results that you need to turn in are:

1. The finite element model (showing the elements, boundary conditions, and applied loads) used for the circular crank and the real crank.
2. Contour plots of the von Mises stress shown on cutting surfaces at cross-sectional planes at 25, 50, and 75% for both the circular crank (xy-plane) and the real crank (yz-plane).
3. Contour plots of the von Mises stress shown on cutting surfaces at longitudinal sections: circular crank (xz-plane at 0 mm), real crank (xy-plane at 50%).
4. Contour plots of the ε_{zz}, γ_{xz}, and γ_{yz} strain components for the circular crank at the cross-sectional plane 125 mm from the centerline of the pedal spindle. Note that strain gage rosettes are located at points *A* (0, -12.5, 125) and *B* (-12.5, 0, 125) on the circular crank. Mark these points (by hand is OK) on your contour plots.

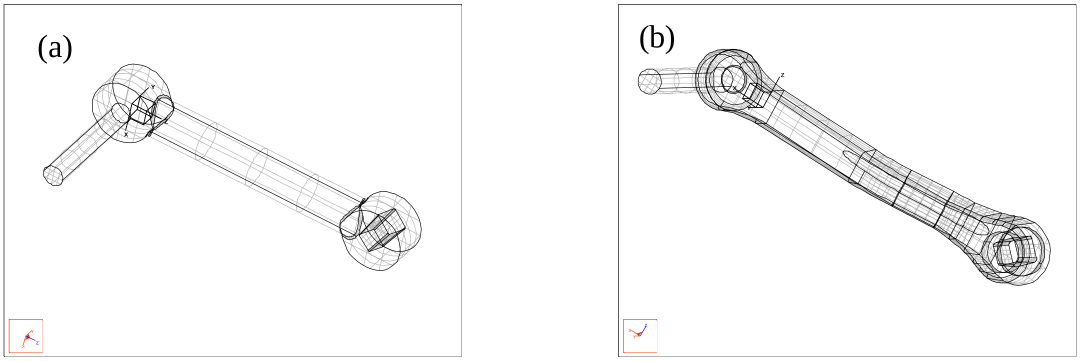


Figure 1: Crank arm solid model; (a) student design and (b) commercial.

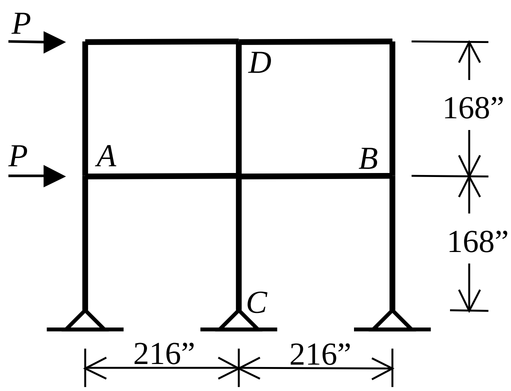
Objectives:

- Understand the differences between solid models and FEA models,
- Understand how FEA fits into engineering design,
- Become proficient in viewing the results of a stress analysis,
- Be able to present FEA results to design engineers.

Attachment 4 – Project 4: Frame Analysis

The concrete frame shown below must resist two equal lateral loads. All columns and beams are 18”×18” square. The columns are pinned at the foundation. The maximum permissible stress is 9 ksi and the maximum permissible lateral displacement is $L/180$ = 1.866 inches. Use E = 5800 ksi and ν = 0.15.

1. Determine the maximum magnitude of the load, *P*.
2. Determine the first 4 natural frequencies.



Turn in the following.

1. Plot of deflected shape with maximum *P* applied.
2. Shear and moment diagrams for members *AB* and *CD*.
3. First four mode shapes.

Objectives:

- Learn how to analysis frame structures using FEA,
- Understand how to scale results of linear FEA,
- Be able to apply multiple design criteria,
- Apply FEA to determine free vibration response.