Integrating Finite Element Analysis into Mechanical Engineering Undergraduate Courses

Shengyong Zhang
Assistant Professor of Mechanical Engineering
College of Engineering and Technology
Purdue University North Central

Abstract

Finite element analysis (FEA) is a powerful analysis tool for simulating the behaviors of a part or system and has been widely implemented in engineering education. To become more competitive in the workplace after graduation, engineering students should have knowledge of the basic FEA concepts and realize the capabilities and limitations of FEA. However, most engineering undergraduate students have little FEA experience and feel unaware of how beneficial FEA can be.

This paper documents an effort to integrate FEA into a series of mechanical engineering undergraduate courses (Mechanics of Materials, Numerical Methods and Vibration Engineering) at the Purdue University North Central. Students learn the fundamentals of FEA mainly through lectures, projects and lab reports. These studies acquaint students with the basic steps in conducting FEA analysis and help students be cognizant of the capabilities of FEA.

1. Introduction

Many engineering problems can be mathematically modeled using differential equations with initial and boundary conditions (Laplace equation for characterizing a heated plate with specified boundary conditions, for example). The differential equations are developed by applying fundamental principles to a system or process (Newton's second law, conservation of energy, virtual work, etc.). Analytical solutions of these differential equations are often useful and provide excellent insight into system behaviors. However, analytical methods work for only a limited class of problems. There are many practical engineering problems for which one cannot obtain exact solutions. Numerical methods are powerful problem-solving tools and provide an alternative for solving those complicated initial-value and boundary-value problems [Chapra, 2014]. There are two common numerical methods employed for solving differential equations: finite difference methods and finite element methods. FEA is a computer based approach for solving a wide range of engineering problems (static and dynamic, steady-state and transient, linear and nonlinear, etc). In contrast to finite difference methods in which all derivatives are replaced by difference equations, FEA methods make use of integral formulations for creating a system of algebraic equations. FEA analysis divides the solution domain into simply shaped regions or elements. An approximate solution for the governing differential equation is derived for each element. The total solution over the entire domain is then generated by assembling the individual solutions, allowing for continuity at the interelement boundaries [Moaveni, 2008].

There have been many efforts to integrate FEA into undergraduate mechanical engineering programs. Six universities collaborated and developed finite element learning models for different undergraduate engineering courses using commercial software [Brown, etc, 2008].

These learning modules provide undergraduate engineering students with new visually oriented insight into the concepts covered in their courses, basic knowledge in finite element theory, and the ability to apply commercial finite element software to typical engineering problems. FEA-based simulations may provide an alternative to some lab experiments which are traditionally performed by hand-on testing. Zhang [2014] documented how students learn to build ANSYS models of their own bicycle frames, calculate the resultant internal forces corresponding to prescribed loads and constraints, and redesign their frame structures to reduce the non-uniformity of the internal force distributions. Moazed etc. [2010] designed a FEA course for undergraduate engineering students, addressing the issues relevant to the practice and use of FEA. The course introduces the concepts of FEA to students in the Strength of Material course during the sophomore year and again in the Machine Design course during the junior year. Due to the complexity of the underlying theories upon which the FEA methods are based, however, a comprehensive study of FEA is not included in the mechanical engineering program curriculum at the PNC.

This paper documents an effort to integrate FEA analysis into a series of mechanical engineering undergraduate courses: FEA is introduced in the Mechanics of Materials course with an emphasis on the main conceptual steps associated with FEA modeling. A beam consisting of an assembly of thin sheets has been modeled using ANSYS for calculating the strains at designated positions caused by external loads. A loaded truss has been simulated in the Numerical Methods course for determining the resultant internal forces in all the truss members. Dynamic FEA analysis has been conducted in the Vibration Engineering course to study an automobile suspension system, and investigate the effects of changing suspension parameters (stiffness and damping coefficient) on the automobile responses to external excitations. The objective of this endeavor is to help students understand the FEA fundamentals and realize the capabilities and limitations of FEA. It is also revealed from informal student surveys that the FEA visual features enable them to visualize analysis results, enhancing their comprehension and retention of lecture materials (vibration modes, for example).

2. Static FEA Analysis

2.1 Stress analysis of a thin-walled beam

FEA is first introduced in the Mechanics of Materials Lab in the junior year. Experimental stress analysis or strain measurement aids engineers in their product designs. On the other hand, FEA has been widely applied for evaluating the strain and stress distributions of a loaded part. A simulation test based on FEA is designed in this lab course for students to learn how to build ANSYS model of the thin-walled beam shown in figure 1 and predict the strain values at designated positions under prescribed loads.

The beam in figure 1 is 600 mm long with a closed-hat section and is assembled using electrical resistance spot welds with a pitch of about 60 mm. Both ends are split longitudinally at the corners and bent outward into mounting flanges to fix the beam in a cantilever mounting at one end and secure a rigid load plate onto the other. All bending radii are held to 1.59 mm.

This beam can be used for conducting experimental stress analysis based on strain gauging technique. As seen in figure 1, eight general purpose 45^o single-plane constantan strain gage rosettes from Measurements Group, Inc. (CEA-06-125UR-120; 3.18-mm gage length) are bonded to the beam at designated positions – where critical stress are likely to occur. After mounting the test specimen to the wall fixture, we load the specimen with an appropriate dead weight. The mounted strain gauge rosettes experience the same stains as the surface due to which the resistance of each gauge changes. The strain signal from each rosette is amplified, filtered, and processed using a LabVIEW virtual instrument (VI) (LabVIEW is a software package from National Instruments Corporation for measurement and control applications). The VI is used to measure the strain values for each rosette. One component of the VI is a half bridge with a resolution of 1.0×10^{-6} and the VI readings can be converted to the experienced strain values, see figure 2.

In this study students learn the main basic steps associated with FEA analysis and how these steps relate to the physical system being modeled (e.g., the specimen, fixture, loading mechanism, etc). Students build their ANSYS models by

- collecting all necessary data, including cross-sectional dimensions, material properties, etc.;
- choosing appropriate finite elements for modeling sheet metals and spot welds;
- defining material models;
- applying a force equivalent to the load applied at the free end in the testing and constraining the other end of the beam properly;
- solving the FE model; and
- recording the strain values at the positions where the gauge rosettes are mounted.

Agreement between the experimentally derived and FEA derived values is good for flange-mounted rosettes 1 – 6. The locations of the two web-mounted rosettes 7 and 8 are near the neutral axis of bended beam, resulting in a low signal-to-noise ratio and precluding accurate strain measurements.

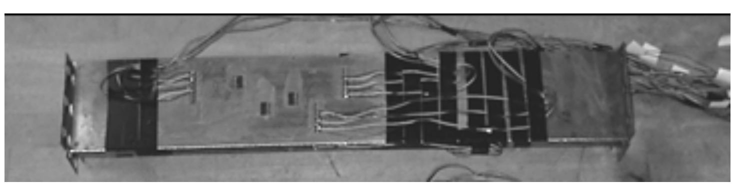

Figure 1 Beam with strain gage rosettes mounted at designated locations. Jumper wires are applied between the gauge tabs and the terminal tabs. Specimen is made of sheet metal of A-366 (commercial grade). The mechanical properties are: yielding strength σ_Y =200 MPa, Young's modulus E=200 GPa, and Possion's ratio ν=0.3.

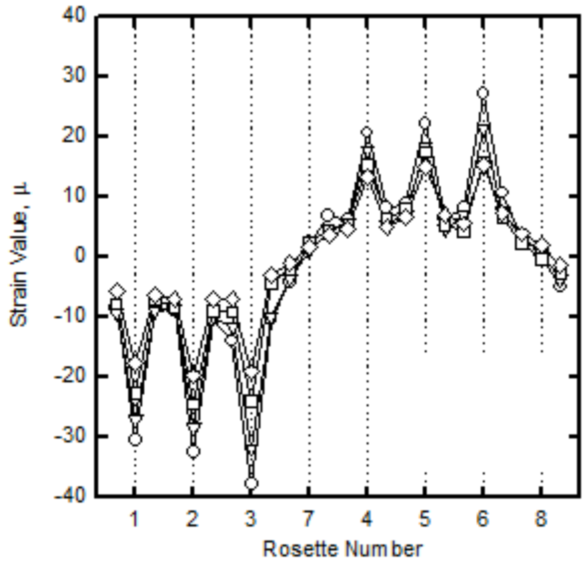

Figure 2 Rosette strain readings. The data point at the rosette numbers correspond to the axial gauges, the points just to the left of the numbers correspond to the left ($+45^o$) gauge, and the points just to the right of the numbers correspond to the right (-45^o) gauge.

2.2 Resultant internal force analysis of a truss

The finite element method (FEM) is one of various methods (numerical integration and differentiation, Runge-Kutta method, finite difference method, etc.) discussed in the Numerical Methods course during the senior year. Because a comprehensive description is beyond the scope of this course, a general introduction of the underlying concepts and theories relevant to FEM is provided in two lecture periods to make students comfortable with the approach. The first class introduces the concepts associated with the derivation of element equations and assembly of individual solutions, which help students understand the truncation errors involved in FEA. In the second class an example based on a steady-state heated rod is employed for demonstrating the major aspects of FEA. Finally, a truss composed of seven pin-connected members has been studied and simulated, see figure 3. Although being simple, this project acquaints students with the following basic steps in conducting FEA analysis:

- using consistent units;
- building geometric models;
- selecting proper element types;
- defining material properties, real constants and cross section properties properly;
- meshing the geometric model;
- applying constraints and loads;
- solving the FE model; and
- post-processing the FE results.

The FE derived internal forces in all the members are compared with those derived from corresponding analytical analysis (Table 1). The FE and analytical values agree well.

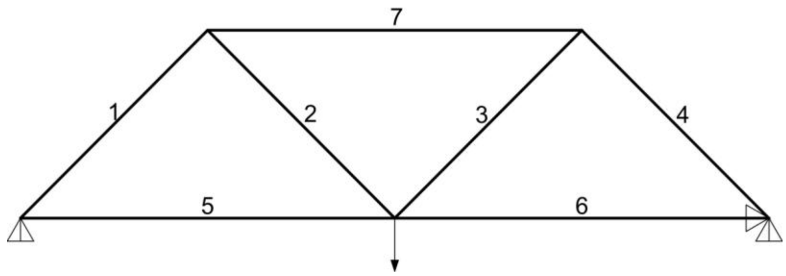

Figure 3 Finite element model of a truss.

Table 1 Comparison of the resultant internal forces in all the members derived from FEA and analytical analysis.

Element Number	1	2	3	4	5	6	7
FEA (N)	-1237.4	671.75	742.46	-742.46	-1050.0	875.0	825.0
Analytical (N)	-1237.6	671.9	742.6	-742.6	-1050.0	875.0	825.0

3. Dynamic FEA Analysis

FEA has been widely employed to solve engineering vibration problems. Integrating FEA into teaching is an efficient way to assist students in learning vibration analysis. Animations and graphical plots enable students to visualize the phenomena of vibrations, enhancing their comprehension of physical insights of engineering vibration.

An automobile suspension system has been studied as a project in this class. One objective of the project is to help students develop ability to apply dynamic FEA skills to simulate spring-mass-damper systems. The main functions of a suspension are usually simulated by spring and damping components which provide the necessary ride isolation at each wheel. Figure 4 shows a bicycle-car model. The sprung mass (M) represents the mass of a vehicle supported on the suspension and the unsprung mass (m) is defined as the total mass of the parts being connected to the wheel directly.

Students simulate the bicycle-car model as a four degree-of-freedom (DOF) system. Impulsive excitation caused by speed bump is chosen to be the input to the suspension in this project. From the ride quality perspective, designers are mostly interested in the vibration of the sprung mass, which is chosen as the system output in this project. The dynamic responses of the sprung mass due to the prescribed excitations are plotted in figure 5.

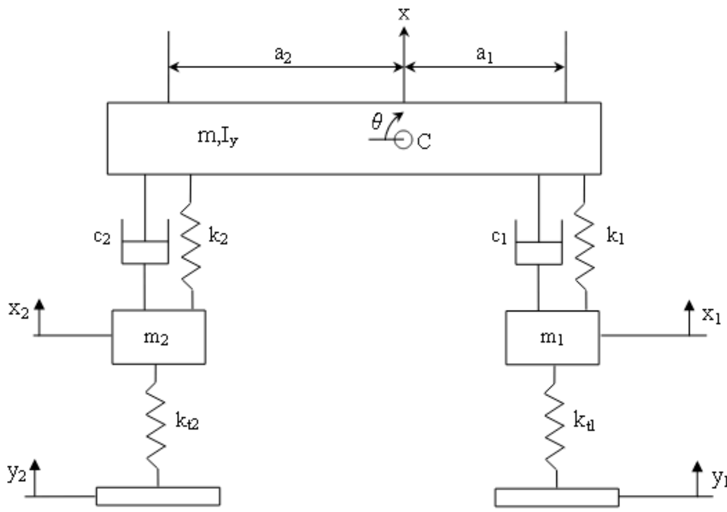

Figure 4 A bicycle-car model having four degrees of freedom, i.e., translational and rotational motion of the sprung mass (bounce and pitch), translational motion of the front wheel, and translational motion of the rear wheel.

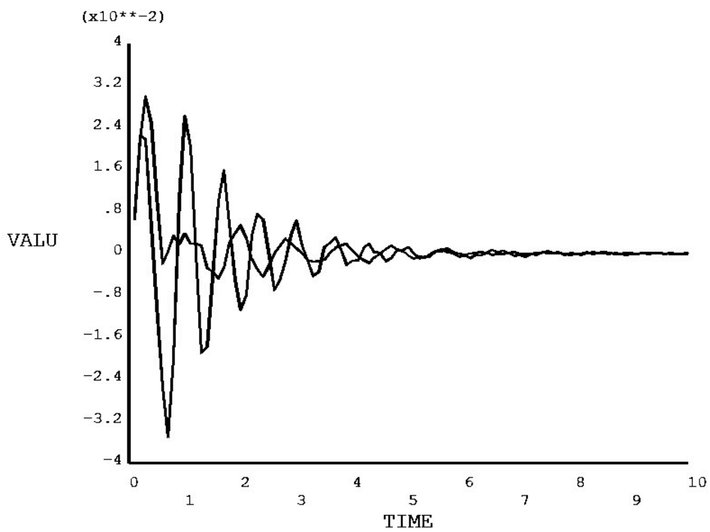

Figure 5 Bounce and pitch responses of the sprung mass due to impulse excitations.

4. Conclusion

It is believed that undergraduate engineering students should have knowledge of the basic FEA concepts and realize both capabilities and limitations of FEA. This paper documents an effort to integrate FEA into a series of mechanical engineering undergraduate courses. Students learn to perform FEA analysis for a thin-walled cantilever beam in the Mechanics of Materials course, a loaded truss in the Numerical Methods course, and an automobile suspension system in the Vibration Engineering course. All these studies help students understand the FEA fundamentals and the student feedback to this effort is positive.

Reference

1. Chapra, S. andCanale, R., 2014, Numerical methods for engineers, 6^{th} edition, McGraw-Hill Higher Education.
2. Moaveni, S., 2008, Finite element analysis: theory and application with ANSYS, 3th edition, Prentice Hall.
3. Brown, A., Rencis, J.J., Jensen, D., et al, (2008). Finite element learning modules for undergraduate engineering topics using commercial software. *ASEE Annual Conference & Exposition*, Pittsburgh, PA. June 22- 25.
4. Zhang. S.Y., 2014, Development of mechanics of materials lab with an emphasis on real-world engineering problem solving, Proceedings of American Society for Engineering Education Illinois – Indiana Section Conference, Terre Haute, IN., March 8, 2014
5. Moazed, A.R., Roberts, R., Le, X. and Duva A., 2010, Teaching finite element analysis in undergraduate technology curriculum. ASEE Northeast Section Conference, Wentworth Institute of Technology, Boston, MA, May 7-8.

Biographical notes: Shengyong Zhang (syzhang@pnc.edu) is an Assistant Professor of Mechanical Engineering at the Purdue University North Central. He has teaching and research interests in the areas of system dynamics, structural finite element analysis, computer modeling and simulation, machine component design, and vehicle body structure optimization.