Paper ID #27931

Incorporating Finite Element Analysis-based Projects in Teaching Machine Component Design

Prof. Shengyong Zhang, Purdue University Northwest

Shengyong Zhang (syzhang@pnw.edu) is an Associate Professor of Mechanical Engineering at the Purdue University Northwest. He has teaching and research interest in the areas of computer modeling and simulation, vibration and acoustics, system dynamics, and automobile lightweight design.

Incorporating Finite Element Analysis Based Projects in Teaching Machine Component Design

Shengyong Zhang
Department of Mechanical and Civil Engineering
College of Engineering and Sciences
Purdue University Northwest

Abstract

Finite element analysis (FEA) has wide applications in industries as a powerful tool for engineering modelling and simulation during the product development process. FEA has also been adopted in the teaching of various mechanical engineering courses. Integrating FEA into the machine design course provides students with a visual insight into the concepts covered in class discussions, and an opportunity to learn the capabilities and limitations of FEA.

This paper documents an effort to integrate FEA into the machine design course at the Purdue University Northwest. Students learn to build various FE models in the assigned class projects. The FEA visual features enable students to visualize the resulting stress and strain distributions, enhancing their comprehension and retention of lecture materials. These FEA-based projects also help students understand the capabilities of FEA and how beneficial FEA can be in the machine design and optimization process.

1 Introduction

Hand-on testing is recognized as an irreplaceable learning experience in the engineering education, but it is often subject to limited availability because of scarce funding for purchasing, maintaining and updating instruments and software. Increasing enrollment of students makes it more difficult to run these tests efficiently. Computer simulation has a variety of benefits when being applied in engineering education [Clark and DiBiasio, 2007]. With increasing sophistication and visual enhancements, computer simulation provides an alternative to some lab experiments which are traditionally performed by hand-on testing. One of the most attractive characteristics in using computer simulations is that students can vary design variables and observe corresponding effects on the results.

Among a wide range of engineering applications in industry (structural analysis, fluid mechanics, heat transfer, acoustics, magnetics, etc.), FEA displays its unique abilities in simulating the performance of a mechanical part or system prior to building a physical prototype. FEA has dramatically driven the machine design path at a much faster pace by, for example, evaluating the stress and deflection of a part under operating conditions, as well as extreme situations which can be difficult or costly to test. Mechanical engineers will most likely find their company has and use FEA in the design of their products and machines.

FEA theories rely upon rigorous mathematics and FEA courses are usually offered as upper-level elective courses in the mechanical engineering curriculum [Chapra, 2014]. To be competitive in the professional field, however, mechanical engineering students should learn the FEA fundamental aspects and be cognizant of the capabilities of FEA. There are a number of commercial FEA software packages (ANSYS, for example) which make the learning process relatively easy.

There have been many publications on the integration of FEA into the machine design course [Richard, 2015]. Moazed etc. [2010] 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, addressing the issues relevant to the practice and use of FEA. 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 theoretical concepts, basic knowledge in finite element theory, and the ability to apply commercial finite element software to typical engineering problems. Wendy [2018] develops five FEA modules in the teaching of machine design course to expose students to interpretation of FEA results. It is concluded that the FEA activities helped students understand the machine design content much better than just performing book problems.

This paper documents an effort to integrate a series of FEA learning modules in the machine design course. A loaded 2D truss has been simulated for learning the FEA fundamentals. The FE model of a cantilever help students investigate the effects of changes in the cross-sectional dimensions on the maximum stress and deflection of the cantilever. FE simulation of a plate with a central hole allows students to visualize the stress concentration due to changes in geometry. FE modelling of an automobile frame exposes students to real world engineering problems. Students learn how to improve frame architecture for stress and deflection reduction. These FEA-based projects are expected to prepare students better for professional engineering careers.

2 FEA-Based Projects

2.1 Rod Finite Element Model of a 2D Truss

A truss is a structure composed of slender members joined together at their end points. There are two assumptions adopted in the design and analysis of trusses. One assumption is that all loadings are applied at the joints and the other one is that all members are joined together by smooth pins. Because of these two assumptions, each truss member will act as a two-force member, and therefore the force acting at each end of a member will be directed along the longitudinal direction of the member. The force in a member is defined to be tensile force if it tends to elongate the member. Otherwise, it is a compressive force. Figure 1 shows the configurations of the truss assigned in this project.

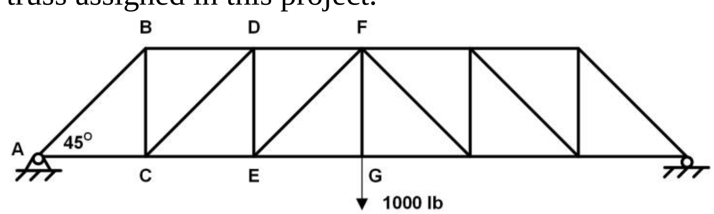


Figure 1. Configurations of a simply supported plane truss. A force of 1000 lb is applied at node G. Material properties and cross-sectional dimensions are defined by students.

Students conduct hand calculations first for the reactions at each support and the internal force in each member. This analytical analysis is based on the joint method and the analysis procedure is as follows:

- Drawing the free-body diagram of the entire truss, building equilibrium equations and solving these equations for the external reactions at all supports.
- Drawing the free-body diagram of a joint having at least one known force and at most two unknown forces, resolving each force acting on the joint into x and y components, building the two force equilibrium equations $\Sigma Fx=0$ and $\Sigma Fy=0$, and solving equations for the unknown member forces.
- Continuing to analyze each of the remaining joints for other unknown member forces.

Students then build FE model of this truss by using rod elements (link180 in ANSYS). Students are expected to learn the following fundamental aspects of FEA through this project:

- Using consistent units.
- Building geometric models.
- Selecting proper element types.
- Defining material properties, real constants and cross section properties.
- Meshing the geometric model.
- Applying constraints and loads.
- Solving the FE model.
- Post-processing the FE results.

Students compare their FE results to those derived from hand calculations, as shown in Table 1. The FE and analytical values agree well. This project helps students realize that FEA is an efficient analysis method and has been a powerful tool in machine design.

Table 1 Comparison of the resultant internal forces in all members.

Truss Member	AB	AC	BC	BD	CD	CE	DE	DF	EF	EG	FG
FEA (lb)	707	500	500	500	707	1000	500	1000	707	1500	1000
Analytical (lb)	707	500	500	500	707	1000	500	1000	707	1500	1000

2.2 Beam Finite Element Models

A beam is defined to be a slender member supporting loads that are applied perpendicular to its longitudinal axis. Beams are widely used elements in various engineering systems, such as I-shaped supporting beam, airplane wings, automobile frames, etc. Beam analysis is an important topic in the machine design course. Students are expected to know how to draw internal shear force and bending moment diagrams, and how to calculate resultant stresses and deflections of a beam subjected to specified constraints and loads.

In this project students simulate cantilever beams and automobile frame structures by using beam element-based FE models. The former has analytical solutions and students can compare their FE results with corresponding analytical ones. The later exposes students to a real-world engineering design problem which helps students learn how to identify critical locations on the auto frames for architecture optimization based on resulting Von Mises stress distributions.

2.2.1 Cantilevered Beams

A cantilever is a beam that is clamped rigidly at one end and free at the other end. Material properties, cross-sectional dimensions, and external loadings have been defined for the cantilever in this project. First, students conduct hand calculations for the maximum bending stress and maximum lateral deflection of the cantilever. Secondly, students build FE model of the cantilever using beam elements (beam188 or beam189 in ANSYS) and compare FE results with those from hand calculations. Finally, students make use of the FE model to learn the changes in the cantilever responses (stress and deflection) due to changes in the cantilever properties (cross-sectional dimensions, material properties, etc.). The following assignments are included in this project:

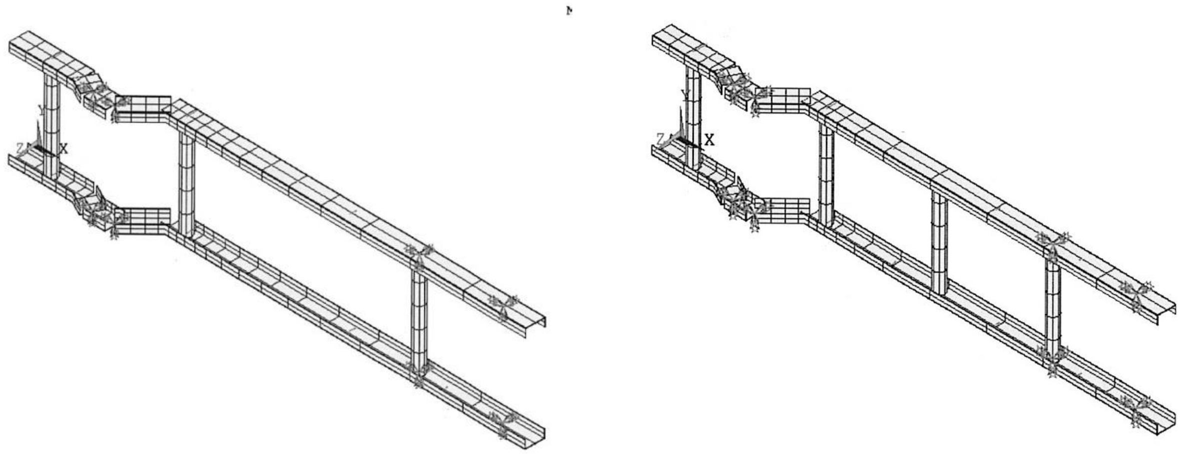
- Compare FEA results of the stress distributions on the cross section at its fixed end with those from hand calculations.
- Compare FEA results of the maximum deflections at its free end with those from hand calculations.
- Change beam cross sections (e.g., from rectangular to round section, from solid to hollow section, etc.) while maintaining constant beam weights. Investigate the relationship between the area moments of inertia and the maximum resultant stresses and beam deflections.
- Maintain constant beam weights and study the effects of material substitutions (for example, steel is replaced with aluminum alloy or magnesium alloy) on the resultant stresses and beam deflections.
- Make use of ANSYS visual features (graphical plots, contour plots, for example) to visualize stress changes along the longitudinal and transverse direction, respectively.

Students have learned the limitations of FEA from this cantilever project. In their FEA models, students constrained all six DOFs at the fixed end of the cantilever according to its definition. This simulation of the clamped end will introduce error in the FEA results, because any fixture has some degrees of flexibility.

2.2.2 Automobile Frames

Mechanical components in the form of simple bars or beams as in the above two projects can be analyzed analytically for closed-form solutions by using basic methods of mechanics. Actual structures, such as automobile frames, are rarely so simple, and it is impossible to build equations for exact solutions. Accordingly, experimental and numerical methods are sought to provide approximate solutions with acceptable errors. FEA is one efficient numerical method for solving ordinary and partial differential equations with specified boundary conditions. It divides the solution domain into simple shaped regions, or elements, and provides approximate solutions by simulating systems with irregular geometry and unusually boundary conditions.

In this project students measure the dimensions of a real light-duty truck frame and built corresponding FE model using finite beam elements, as shown in Figure 2(a). The FE model is composed of two curved beams with C-shaped cross section along the longitudinal direction and three cross members along the transverse direction. Cross members help enhance structural stiffness to avoid excessive deformation under external loads. Students constrain the frame at the four suspension/frame mount locations. Parallel uniformly distributed loads are applied vertically on both sides. Based on the resulting Von Mises stress distributions of the frame under the applied constraints and loadings, students continue to modify the frame architecture by introducing another cross member at appropriate locations for strength and stiffness reinforcement, see Figure 2(b).



(a) (b)

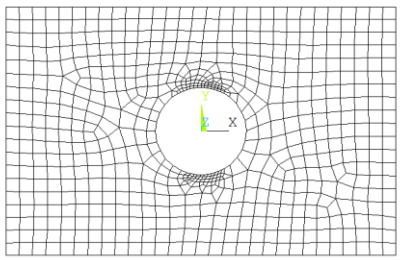
Figure 2 (a) FE model of the original light-duty truck frame; (b) FE model of the reinforced truck frame by introducing more cross members at appropriate locations.

This project challenges students to adopt advanced ANSYS functions in their FE models. For instance, students have learned to define the orientations of beam elements in the auto frame models. Beam elements are used to create a mathematical one-dimensional idealization of a 3-D structure. A beam element is represented as a line in FE models, without inherent orientation of its cross-sections. Students should specify corresponding orientation for each beam element in their FE models. Through this project, students understand the capabilities of FEA and how beneficial FEA can be in the product design and optimization.

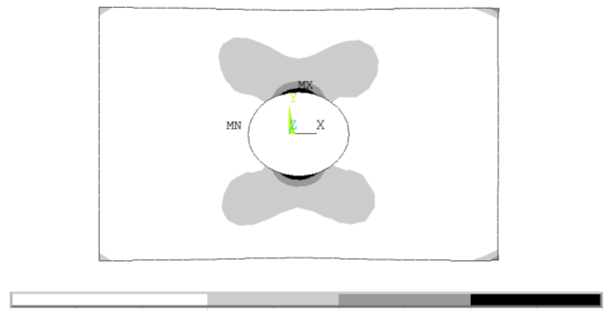
2.3 Plate Finite Element Models

Plate theory is out of scope of this machine design course. This project is designed to provide students with a visual insight into stress concentrations. Stress concentration occurs in a loaded object where there is an abrupt change in geometry, resulting in localized stress which is considerably higher than nominal value. Many publications make use of a rectangular plate with a central circular hole to demonstrate the stress concentration mechanism. In this project, it is assumed the plate is constrained at one side and an evenly distributed force is applied at the opposite side. Students choose appropriate dimensions of the plate.

Both analytical analysis and FEA analysis are conducted in this project. First, each team works on the constrained and loaded plate for analytical solutions. Students conduct hand calculations to determine the stress concentration factor based on predefined dimensions and calculate corresponding maximum stress at critical locations. Secondly, students build FE model by using finite plate elements. Figure 3(a) show the plate FE model with the left side constrained and the right side uniformly loaded. Stress concentration is obvious in the resulting Von Mises stress distribution as shown in Figure 3(b).



(a)



(b)

Figure 3 (a) FE model of a rectangular plate with a central hole; (b) maximum Von Mises stress in the vicinity of the hole.

3. Student Assessments

Follow-up survey was conducted for assessing the efforts of integrating the FEA-based projects in the machine design course. The four questions in the survey are as follows:

1) FEA-based projects help me understand the concepts discussed in the machine design course.
2) FEA-based projects help me improve my critical thinking in machine design considerations.
3) FEA-based projects help me understand the capabilities and limitations of finite element method.
4) FEA-based projects improve my techniques and skills of FEA modelling for engineering practice.

Responses to the first question show that students benefit from these FEA modelling projects, which provide a new visually oriented insight into the theoretical concepts. The second question is one of the author’s teaching goals that foster critical thinking and life-long learning to promote continuing professional development. Responses to the third survey question indicate that there are approximately one third of students who are not clear about the capabilities and limitations of FEA. This is mainly because (1) students have no experience in FEA modeling before taking this course; and (2) students are short of the theoretical background to understand the numerical results from FEA. The author believes that an brief introduction to the FEA fundamentals and more real life engineering projects (airplane wings, for example) will be beneficial for students to realize that FEA is an efficient and powerful tool in machine designs. The last question measures the contribution of this innovative teaching effort to one of the student outcomes in ABET Criterion 3 which emphasizes on the ability to use the techniques, skills, and modern engineering tools necessary for engineering practice.

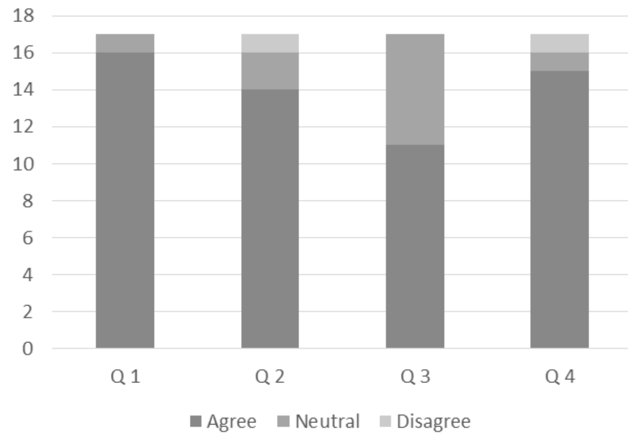


Figure 4 Student assessments of the FEA-based projects on their learning

4. Conclusion

It is believed that FEA-based computer simulation has a variety of benefits when being applied in engineering education and research. This paper documents an effort to integrate FEA modeling into Machine Design course teaching. Students learned to build a series of FE models in ANSYS and utilize these models to enhance their understandings of the strength and stiffness analysis discussed in class. Additionally, students are exposed to real-life engineering problems and learned how to improve structural performance based on FE results. Student feedback to this effort is positive.

Reference

1. Clark, W. and DiBiasio, D. (2007). Computer simulation of laboratory experiments for enhanced learning. ASEE Annual Conference and Exposition. Honolulu, Hawaii, June 24-27.
2. Chapra, S. and Canale, R., 2014, Numerical methods for engineers, 6th edition, McGraw-Hill Higher Education.
3. Richard G Budynas and J Keith Nisbett, 2015, Shigley's Mechanical Engineering Design, 10th edition, McGraw-Hill Higher Education.
4. 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
5. 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.
6. Wendy S. Reffeor G, 2018, Using FEA as a Pedagogical Tool for Teaching Machine Component Design. 2018 ASEE Annual Conference & Exposition. Salt Lake City, UT., June 24-27.

Biographical Notes

Shengyong Zhang (syzhang@pnw.edu) is an Associate Professor of Mechanical Engineering at the Purdue University Northwest. He has teaching and research interest in the areas of computer modeling and simulation, vibration and acoustics, system dynamics, and automobile lightweight design.