Session #: _____________

UNDERGRADUATE RESEARCH ON IMPROVING ASSEMBLY EFFICIENCY OF SPOT-WELDED STRUCTURES BASED ON FINITE ELEMENT METHOD

Shengyong Zhang

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

1. ABSTRACT

Finite element method (FEM) shows its great power in the mechanical product design by simulating the performance of a part or system prior to building a physical prototype. It has been widely employed to solve problems relating to structure modeling and analysis, heat and mass transfer, fluid flow, etc. Most mechanical undergraduate students, however, do not realize how beneficial FEM can be, because it is usually taught at the postgraduate level.

This paper documents the issues relevant to introduce FEM to mechanical engineering students in the Engineering Analysis course during the senior year. Besides a brief introduction of the general step-by-step procedure involved in a typical finite element analysis (FEA), a research is conducted on improving assembly efficiency of spot-welded structures using ANSYS. This research serves as a practical application to make students to be cognizant of the capabilities of FEA.

Key words: spot welding, assembly efficiency, finite element.

2. INTRODUCTION

The FEM or FEA is a numerical technique and computer based approach for simulating the performance of a part or system prior to building a physical prototype. Few tools have shown such a great power for product design, assessment and optimization. It has dramatically driven the engineering design path at a much faster pace by, for example, evaluating the stress level and displacement behavior of a part under operating conditions as well as extreme situations which can be difficult and costly to test, or identifying the potential failures and cost reductions early in the preliminary design stages. More and more employers expect bachelor-level engineering graduates to learn and apply this well-established technique. To be competitive in the professional field, a mechanical engineer cannot afford to be blind to the capabilities of FEA associated with building finite element models, solving the resulting equations and displaying the results for examinations. However, the FEA courses are usually offered at the postgraduate level, partially because of its heavy dependence on a rigorous mathematical background.

Different approaches have been taken to integrate FEA into undergraduate engineering programs. Moazed etc. [2010] introduce the concepts of FEA to BMET students in the Strength of Material course during the sophomore year and again in the Machine Design course during the junior year. The formal introduction to FEA is offered in the last semester of the students' curriculum. Mueller [2003] makes use of MATLAB to introduce FEA to mechanical engineering sophomore students. The orderly finite element solution procedure is emphasized and illustrated with a statically indeterminate problem that is solved both by hand calculation and MATLAB program. 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.

The FEM is one of various numerical methods (curve fitting, finite-difference method, for example) discussed in the Engineering Analysis course [Chapra, 2010] offered to senior mechanical engineering students. The general step-by-step procedure involved in a typical FEA is presented in two lecture periods. A research is conducted on improving assembly efficiency of spot-welded structures by using ANSYS. This research serves as a practical application to make students to be cognizant of the capabilities of FEA.

2.1 Class Lecture

Finite-difference methods solve partial differential equations (PDE, Laplace equation for characterizing a heated plate with specified boundary conditions, for example) by dividing the solution domain into a grid of discrete points or nodes, writing the PDE for each node, and replacing its derivatives by finite-divided difference. In contrast to finite-difference techniques, the FEM divides the solution domain into simply shaped regions or elements and provides a better way for simulating systems with irregular geometry, unusually boundary conditions, or heterogeneous composition. However, a comprehensive description of the FEM is out of scope of this course. Class lecture focuses on the general overview of the steps involved in a typical FEA, followed by two simple examples for demonstrating major aspects of FEA.

The first class introduces the concepts associated with element equations (i.e., the shape functions and interpolation functions) and the method of weighted residuals for developing element equations. This derivation process is crucial for students to understand the truncation errors involved in FEA. The entire system equation is the direct result of assembly, based upon the assumption that the solution for continuous elements is matched so that the unknowns at their common nodes are equivalent. The resulting system equation is usually expressed in a banded matrix form in which all elements equal to zero, with the exception of a band centered on the main diagonal. Various techniques for solving linear algebraic equations (i.e., partial pivoting, LU decomposition, Gauss-Seidel method) help students understand how to yield solutions at all nodes.

In the second class, two simple examples are employed for demonstrating the major aspects of the FEM. The first one is for modeling a steady-state, one-dimensional heated rod. A constant heat source is defined along the rod and the ends of the rod are held at fixed temperatures. The rod has a length of L and is divided into four equal-length elements. A system of five equations is generated by following the above-mentioned procedure. Solving the system of equations by hand will yield the solution at each node on the rod. The other example is a loaded truss composed of seven pin-connected members. Some members are made of carbon steel while the others are made of aluminum alloy. Link element in ANSYS is selected for modeling the truss, see Figure 1. Although being simple, this example shows students the ways to select the proper element types, apply constraints and loadings, defined material properties, and use consistent units. The internal forces in all the members are calculated and compared with corresponding analytical solutions. The results are consistent with each other.

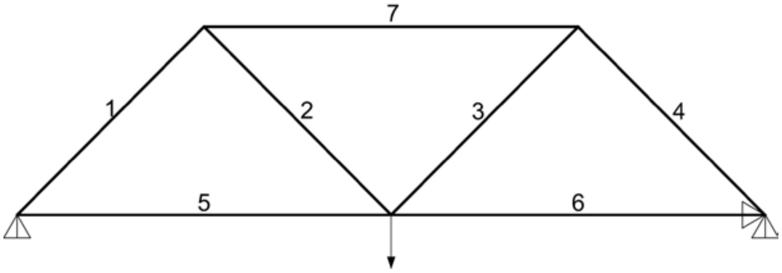


Figure 1: Finite element model of a truss

2.2 Research Project

Spot welding has been widely used in various engineering applications because of its high speed, energy efficiency, lack of work piece deformation, and adaptability for automation. The integration of spot welds determines the overall assembly stiffness of welded structures and much research has been performed on the properties of spot-welded joints. Rudy et al. [1956] studied the strain distribution around spot welds and suggested that the weld diameter to sheet thickness ratio is the most important parameter for determining the strength of a spot weld. Hess et al. [1945] concluded with two types of fatigue failure of spot-welded joints under tensile-shear loading. Deng et al. [2000] investigated the mechanical behavior of spot welds using three-dimensional FEA and provided a mechanics-based explanation for experimental observations reported earlier in the literature.

Spot-welded structures provide a good example for students to understand the capabilities and limitations of FEA. First, students can select different types of finite elements in their models. One can build a detailed finite element model by meshing both the spot nuggets and welded metal sheets by using 3-D solid elements. One can also model each sheet using general shell elements and each spot weld by two sets of radial beam elements lying on each sheet and a link element connecting the two sets at the center of the spot nugget. Secondly, students can perform a variety of analyses: linear static analysis under either tensile-shear loading or bending loading; boundary nonlinear analysis with 3-D contact elements applied in the overlap region; topological optimum design for maximizing the assembly efficiency, etc. Finally, it is easy to incorporate some other joining techniques for metal sheets in the finite element models. For instance, students who are interested in this research can modify the finite element models to investigate the mechanical characteristics of weld-bonded structures (spot welding combined with adhesive bonding). Figures 2 and 3 show a specimen and corresponding finite element model.

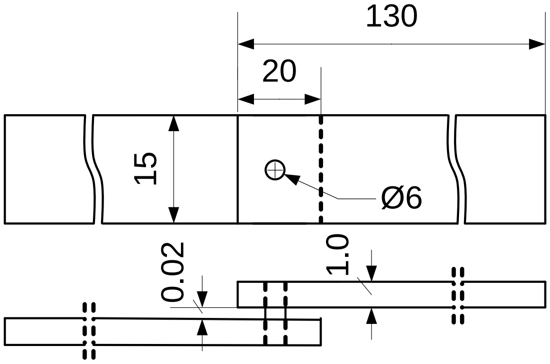


Figure 2: Spot-welded structure. The mechanical properties are: Young's modulus = 190GPa and Possion's ratio = 0.25 for sheets; Young's modulus = 200GPa and Possion's ratio = 0.2 for welds. All dimensions are in millimeter.

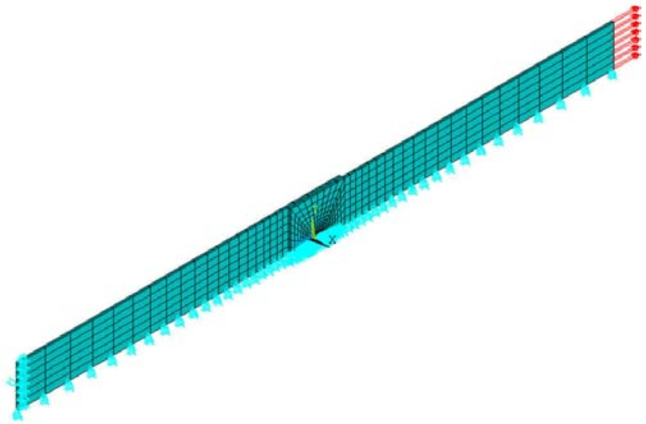


Figure 3: Finite element model. One end of the specimen is fixed on a rigid wall. The other end is subjected to a uniformly distributed loading. Symmetry boundary conditions are applied to represent the symmetrical parts.

Static finite element analysis results in unevenly distributed von Mises stress around the spot nugget as shown in Figure 4. Material under high stress is usually considered to be more effective on the material usage. Based on this, students may take a further step to perform topological optimization analysis in which the structural compliance is selected as the objective function while the volume of the material in the overlap region of the specimen is defined as constraint. The optimum result (Figure 5) shows that the stress away from the spot nugget is extremely small compared with that around the nugget, indicating the inefficient use of material in this region. For lightweight design, material can be removed from this less efficient region with little negative influence on the structural performance.

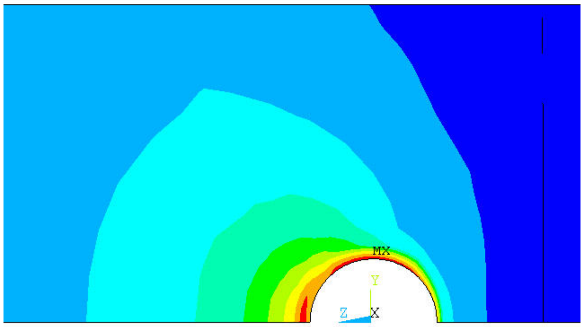


Figure 4: Von Mises stress distribution around the nugget

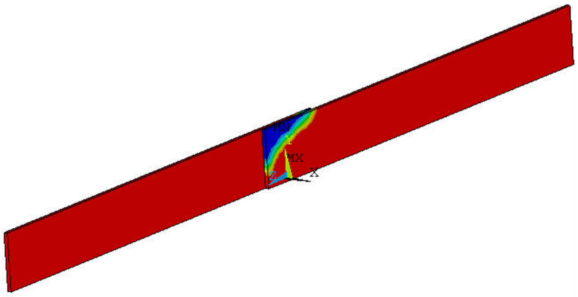
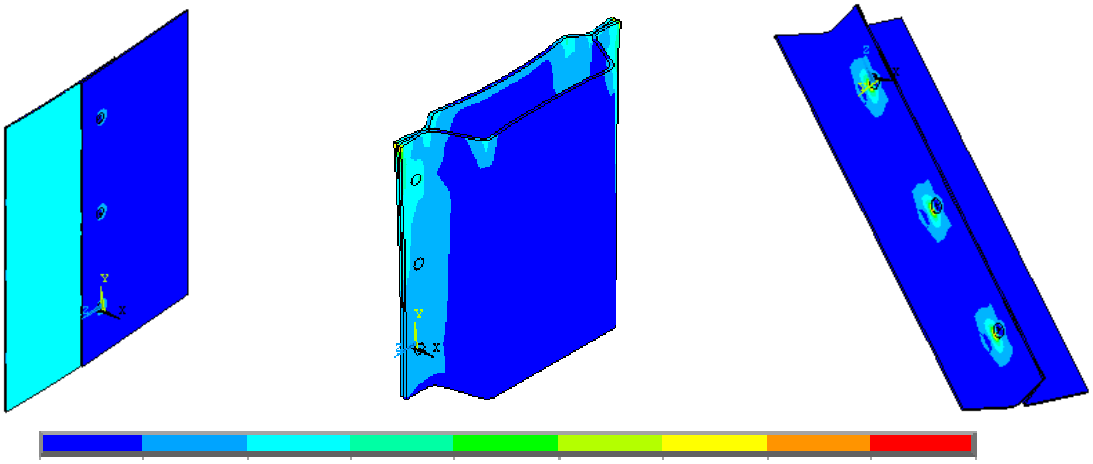


Figure 5: Result from topological optimization

Three kinds of spot-welded structures (lap, double-hat and T-shape) are selected for application studies. Figure 6 shows the finite element results of the three original structures. Three improved structures are derived from corresponding original ones by reducing the material in the less efficient overlap region. The mass of the material in the overlap region has been decreased by about 20% in each improved model. Figure 7 shows the finite element results of the three improved structures. The structural stiffness and the assembly efficiency (the quotient of the structural stiffness and the mass of the material in the overlap region) are calculated for each of the three original and three improved structures. The results are listed in Table 1. It is concluded that the assembly efficiency of spot-welded structures can be improved while maintaining the stiffness by properly reducing the material from the overlap region.

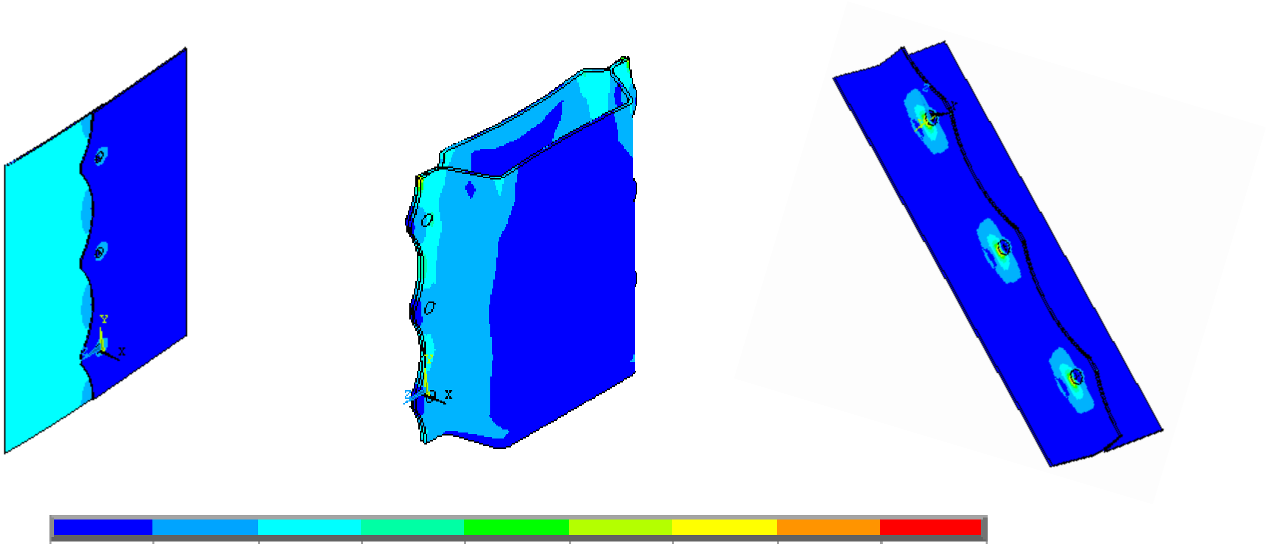
3. CONCLUSIONS

This paper documents the issues relevant to introduce FEA to mechanical engineering students in the Engineering Analysis course during the senior year. Finite element research on spot-welded structures serves to encourage student to learn in an effective and efficient way and make students to understand the capabilities and limitations of FEA. Reactions of students to this research are positive.



(a) lap (b) double-hat (c) T-shaped

Figure 6: Finite element results of the three original structures



(a) lap (b) double-hat (c) T-shaped

Figure 7: Finite element results of the three improved structures

Table 1: The ratios of the structural stiffness and the assembly efficiency of each pair of the improved and original models.

	Lap Structure	Doubt-hat Structure	T-shaped Structure
Stiffness	0.99	0.99	0.97
Assembly Efficiency	1.24	1.31	1.22

REFERENCES

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.

Chapra, S.C. (2010). *Numerical Methods for Engineers*. 6th edition, Mcgraw-Hill Companies.

Deng, X., Chen, W. and Shi, G. (2000). Three-dimensional finite element analysis of the mechanical behavior of spot welds. *Finite Element in analysis and Design.* **35**, 17-39.

Hess, W.F., Wyant, R.A. and Averbach, B. L. (1945). An investigation of the spot welding of Aluminum alloys using condenser-discharge equipment. *Weld. J*. **24**, 404s-412s.

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.

Mueller, D. W. Jr. (2003). Introducing the finite element method to mechanical engineering students using MATLAB. *ASEE Annual Conference and Exposition,* Nashville, TN. Jun 22-25.

Rudy, J.F., McCauley, R.B. and Green, R.S. (1956). The behavior of spot welds under stress. *Weld. J.* **35(2)**, 65-71.