Paper ID #24637

Teaching Finite Element Analysis for mechanical undergraduate students

Dr. Xiaobin Le P.E., Wentworth Institute of Technology

Professor, Ph.D, PE., Department of Mechanical Engineering and Technology, Wentworth Institute of Technology, Boston, MA 02115, Phone: 617-989-4223, Email: Lex@wit.edu, Specialization in Computer Aided Design, Mechanical Design, Finite Element Analysis, Fatigue Design, Solid Mechanics and Engineering Reliability

Prof. Richard L. Roberts, Wentworth Institute of Technology

Associate Professor, Department of Mechanical Engineering College of Engineering, Wentworth Institute of Technology, 550 Huntington Ave., Boston, MA 02115

Prof. Anthony William Duva P.E. , Wentworth Institute of Technology

Anthony W. Duva An Associate Professor in the Mechanical Engineering and Technology Department at Wentworth Institute of Technology since 2001 with 14 years of prior full time industrial experience. He has worked in the design of various technologies from advanced underwater and ultrahigh altitude propulsion systems to automated manufacturing equipment. His interests include advanced thermal and mechanical system design for green power generation.

TEACHING FINITE ELEMENT ANALYSIS FOR MECHANICAL UNDERGRADUATE STUDENTS

ABSTRACT

Since Finite Element Analysis (FEA) has become a daily tool for design engineers in industries, mechanical engineering students must learn this tool during their education. MECH4200-Simulation-Based Design teaches how to apply FEA commercial software for mechanical undergraduate students. Teaching how to use the FEA commercial software is the main contents of this type of course. There are lots of approaches to teaching this type of course. Authors believed that understanding of the fundamental concepts of FEA theory would significantly help students to have a better understanding of FEA theory and to utilize the application of an FEA commercial software. In 2018 spring semester, authors proposed and implemented a teaching approach which included teaching fundamental concepts of FEA theory, teaching its commercial software and implementation of it in class design projects. Although some students complained about the complexity of fundamental concepts of FEA theory and tedious theoretical calculations, 92.3% of students agreed that teaching the fundamental concepts of FEA theory helped them to have a better understanding of the FEA commercial software. 92.3% of students agreed that teaching the fundamental concepts of FEA theory should be kept as part of the course. At the end of the course, we asked students to take the CSWA-S Certification, 60% of students in the section with the proposed approach passed the certification exam while other sections had an average 35.9% of pass rates. Our experience indicated that teaching the fundamental concepts of FEA theory significantly helped students to have a better understanding of FEA application and to facilitate them to use FEA commercial software.

1. INTRODUCTION

No. 1 design criteria for mechanical design is designing safe components. It is well known that theoretical calculations of stress and strain can be conducted only on components with simple geometries under simple loading conditions. But, FEA (Finite Element Analysis) simulation can calculate the stress/strain of components and assemblies with complicated geometries under complicated loading conditions. In the current industry practice of mechanical design, the prototype of design may not be manufactured for testing unless the design has been approved to be safe through FEA analysis because such prototype with failure through FEA simulation has an extremely high possibility of failure and the cost of manufacturing and testing prototype might be very high. Because FEA is a daily tool for design engineers in many industries [1.2], mechanical engineering students must know how to use this tool during their education. Many engineering colleges have offered FEA-related courses to their engineering students [3,4,5,6,7]

FEA commercial software is a version of solid mechanics and is expressed in the form of computer codes. It is commonly accepted that teaching pure theory and programming of the

Finite Element Method (FEM) is for Master-Degree or Ph.D. candidate students and has not been taught for undergraduate engineering students. Even though the FEM theory and FEA commercial software are extremely complicated, most FEA commercial software has user-friendly interfaces and is easy to use. There were many ways to introduce FEA software to students. One of the common ways was to let students gradually learn FEA software, including introducing the FEA software during engineering statics in their sophomore year [8,9], mechanics of materials in their sophomore year [9, 10,11] and design of machine elements in their junior year [12, 13,14]. Finally, a full course in finite element analysis was offered for junior or senior engineering students [1~7].

There were lots of approaches to teaching the FEA course. One frequently-used approach was focusing on teaching the application of FEA commercial software through case studies. In this approach, various important interface settings of the FEA commercial software were explained in detail and practiced through homework and case studies. This was the approach we used in the past. Through direct observations and conversation with students, we found that: some students had difficulty choosing proper settings through the software interfaces; some had no clue why the settings were required, and some had some difficulty properly interpreting the simulation results. They could not clearly explain that reducing element size would increase the accuracy, the error of FEA simulation results might mainly come from boundary conditions (loading and restraints) and the relative errors of the displacement was much smaller than that of the stress in a convergence study. Although these had been clearly explained during lecturing, students did not resonate with the explanations because they didn't understand them. Another frequently-used approach was a combination of the introduction of the fundamental FEA theory and utilization of commercial FEA software. It is a common belief that understanding some fundamental concepts of FEM theory will significantly improve students' ability to apply FEA software in engineering design [4, 5, 15, 16, 17]. In 2018 spring semester, authors adapted this approach to teaching our FEA course. The feedback and results were very encouraged. This paper will present what we did by using this teaching approach.

Our mechanical engineering program offers a required FEA course in the senior year. We strongly believed that understanding the fundamental concepts of FEM theory would significantly help students to have a better understanding of FEA simulation and to facilitate the application of FEA commercial software. In the spring semester of 2018, the teaching approach in this course included three key contents: (1) teaching fundamental concepts of finite element analysis theory, (2) application of FEA commercial software, and (3) implementation of gained skills in class design projects. This paper will discuss and present what we did in teaching an FEA course for undergraduate mechanical students. The main focus presented in this paper will be for teaching fundamental concepts of FEM theory utilized in the first 3 weeks of the semester. At the end of the course in 2018 spring semester, students participated in a class survey and took the CSWA-S certification exam (Certified SOLIDWORKS Associate – Simulation). Analysis of the collected data indicated that teaching the fundamental concepts of FEA theory significantly helped students to have a better understanding of FEA and facilitated them to use FEA commercial software.

2. TEACHING FINITE ELEMENT ANALYSIS FOR MECHANICAL ENGINEERING STUDENTS

2.1 THE BRIEF DESCRIPTION OF MECH4200-SIMULATION-BASED DESIGN

The MECH4200-Simulation Based Design course utilizes lectures, demonstrations and case studies for the teaching how to use FEA commercial software for stress-strain analysis of components and assemblies. It is a required course for senior mechanical engineering students and has 2-lecture hours and 4-lab hours for a total of 4 credits. SolidWorks Simulation is the FEA commercial software for this course. SolidWorks is the computer-aided engineering computer program used in our mechanical engineering program and offers a complete platform for all purposes of mechanical engineering needs such as 3D modeling, drawing, finite element analysis, and computer-aided manufacturing. The mechanical engineering program at Wentworth has 1000 educational licenses to ensure every mechanical student has full access to every module in SolidWorks academic suit. Prerequisite courses teach the use SolidWorks to create models and drawings starting in their freshman year including simple analysis using FEA in courses such as Engineering Statics, Mechanics of Materials and Design of Machine Elements. In Mechanics of Material course during their sophomore year, they conducted three finite element simulation labs [10] in which they treated the SolidWorks simulation as a virtual instrument and used it to simulate and visualize stress/strain of bars under axial loading, shafts under torsion and beams under lateral force and bending. In Design of Machine Elements course in their junior year, they used SolidWorks simulation to simulate and visualize the stress and strain of components with irregular shapes. In these two cases, instructors provide detailed procedures and demonstrate with explained examples, students mainly followed instructors' demonstrations to conduct their FEA simulations.

In the spring semester of 2018, the newly-proposed teaching approach for MECH4200-Simulation Based Design included three key components: (1) teaching fundamental concepts of FEM theory, (2) teaching how to utilize commercial FEA software, SolidWorks Simulation, for stress /strain analysis on components and assemblies and (3) the implementation of SolidWorks simulation for two design projects.

2.2 FUNDAMENTAL CONCEPTS OF FINITE ELEMENT METHOD THEORY

The first three weeks out of a total of 15 weeks in the Simulation-Based Design course was devoted to teaching fundamental concepts of the FEM. The purpose of this was to help students gain a basic understanding of FEM theory and allow students to have a better understanding of the various settings in the FEA software.

The basic concepts or a general procedure for FEM theory [18] can be summarized as followings:

(1) The FEM is a numerical approximation technique for solving deformation of an object. In this technique, after the x, y and z components of the deformation: $u(x,y,z)$, $v(x,y,z)$ and $w(x,y,z)$ of an object are determined, stress/strain of the object can be calculated through applicationthe of Hooke's law.
(2) The object is divided into a lot of small divisions (finite elements).
(3) Each element has several nodes along its edges or on its outer surfaces. The adjacent finite elements are joined by the shared nodes. Therefore, the object is replaced by or represented as an assembly of finite elements connected at the shared nodes.

(4) Approximate deformation or shape functions inside a finite element between nodes are hypothesized and then determined in terms of nodal deformations.
(5) Different techniques such as the method of least squares and the minimum potential energy principle are used to make the errors minimum and thus to convert a continuous physical problem (an object) into a set of linear algebraic equations, which are expressed as a linear function of nodal deformation.
(6) The loading conditions and restraint conditions of the object are applied to the set of linear algebraic equations. Then the set of the linear algebraic equations can be solved to obtain the nodal deformation of each node.
(7) Once the nodal deformations of each node inside an element are known, Hooke's law can be applied to each element using the element's shape functions to calculate stress /strain of any point of the object.

There are 6 key fundamental concepts of the FEM theory are: (1) Types of elements, (2) Approximate deformation or shape functions of an element, (3) Element stiffness property which links together forces, nodal deformations, material properties and element geometry, (4) The global linear algebraic equations in which the shared nodes are joined, (5) Application of the loading and restraints in the global linear algebraic equation, and (6) Stress/strain calculation. The best way to teach and to demonstrate these 6 key fundamental concepts of the finite element method is through simple examples.

Since the FEM theory can be considered extremely complicated for undergraduate students, a simplified treatment was warranted for students to familiarize themselves with some fundamental concepts of FEM theory. The authors collaborated to develop a simplified and straight forward approach to do this in three weeks. It took a long time and lots of work to summarize these examples for teaching the fundamental concepts of FEM theory. These examples will be included and displayed in Appendixes of this paper and might be useful for a reference. In 2018 spring semester, we developed three lectures and three labs to discuss the fundamental concepts of the FEM theory.

- Lecture one was about the various types of elements and how to formulate approximate deformation functions and then how to determine the shape functions.
- The second lecture was about using the minimum potential energy principle to make the errors minimum and thus to link internal forces, deformation, material property, and geometry together and how to form global linear algebraic equations.
- The third lecture provided a demonstration example of a 1D stepped bar under axial loading including determining boundary conditions, stiffness characteristics and all steps required to solve theoretical equations using hand calculations.

After each lecture, we conducted one lab section dedicated to working on examples together with students prior to students working on homework assignments during these faculty-guided labs. In the fourth week, an exam was administered to assess students' understanding of the fundamental concepts of FEM theory. The date for this exam was announced on the first date of the class to ensure students focused on the topics presented in lectures and labs. The following are concise descriptions of the key topics presented to the students for teaching the fundamental concepts of FEM theory.

1) Types of Elements

One of the fundamental concepts of FEM theory is to divide an object into many smaller finite elements. A node in a finite element is a coordinate point which is on the edges or on the outer faces of the elements, which are sketched as a round dot in Figure 1. One element can have multiple nodes but must have at least 2. The variety of element types used in FEA analysis were presented and discussed. For example, a bar under axial loading can be simplified as 1D elements because stress and strain are exactly the same in the same cross-section. For a 2-D plane stress problem, we use 2D elements because the stress and strain are the same in the thickness or depth direction. Sheet metal problems it can be represented by a shell element because the stress /strain in the thickness direction is only due to a bending moment. When considering the general case, the object can be divided into 3D elements. Several element types shown to the students are depicted in Figure 1.

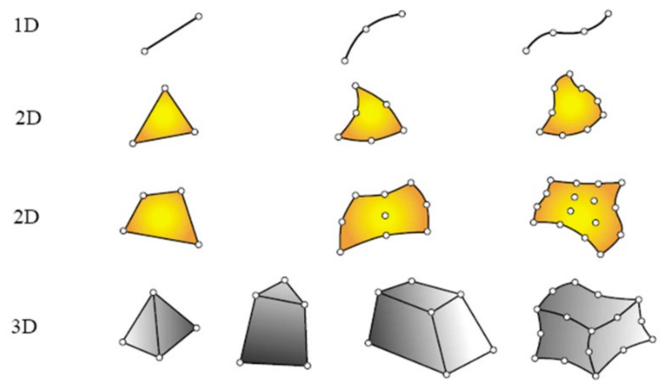

Figure 1 several elements

2) Deformation functions and shape functions

The second of fundamental concepts of FEM theory is to build an approximate deformation function inside an element, which is fully defined by the nodal deflections of the element.
It is very difficult or impossible to build a single deformation function to describe the deformation of an entire object which may have complicated geometries, loading, and restraints. However, when the object has been divided into many small finite elements, we can approximate the element internal displacement by using an appropriate type of deformation approximation such as linear function, second-order function, third order functions and so on depending upon the number of nodes in an element.

Deformation function of a finite element is a continuous function of the coordinates and valid only inside this element and are only solely dependent upon the deformation of nodes inside the element. In FEM theory, we typically use the shape functions to express the approximate deformation functions. In the following, we will only use the x-component of deformation function as an example. The same method can be used for y- and z- component of deformation functions.

$$u(x,y,z) = \sum_{i=1}^{n} N_i(x,y,z)u_i \tag{1}$$

Where $u(x, y, z)$ is the x-component of the deformation at the coordinate point (x, y, z). u_i is the x-component of deformation at the i^{th} node point. n is the number of total nodes of the element. $N_i(x, y, z)$ is the i^{th} shape function for the i^{th} node and is an interpolation function. The shape functions have the following three properties.

$$N_i(x, y, z) = 0 \quad \text{when } i \neq j \text{ at the } j^{th} \text{ node} \tag{2}$$

$$N_i(x, y, z) = 1 \quad \text{when } i = j \text{ at the } j^{th} \text{ node} \tag{3}$$

$$\sum_{i=1}^{n} N_i(x, y, z) = 1 \tag{4}$$

We used four examples during lecturing to explain how to build deformation functions and to explain the shape functions. These four examples are displayed for the reference in Appendix A.

3) The minimum potential energy principle and element properties

FEM can provide a numerical approximate result. Several different techniques such as the minimum potential energy principle can be used in FEM to make the errors minimum. Through the minimum potential energy principle, we can establish the relationship among internal forces at nodes, nodal deformation, material property and element geometry. This relationship is typically called as element properties. We used a 1D element with two nodes during lecturing to show this process. This example is displayed for the reference in Appendix B.

4) Assembling the global equations

Now that the object has been divided into many independent finite elements. It should be emphasized that the shared nodes on adjacent elements are joined together and the shared nodes on adjacent elements have the exact same deformation. So, the FEM does not analyze an original object but analyzes the assembly of many finite elements which are joined through the shared nodes on adjacent elements. We used a 1D-element example during lecturing to explain this process. This example is displayed for the reference in Appendix C.

5) A 1D-element example for calculating stress/strain

The best way to provide students with a better understanding of the fundamental concepts of FEM theory is to manually solve a stress /strain problem by using the FEM theory. We used one lecture to explain and to demonstrate a 1D-element example. This 1D-element example we did during the lecture, is displayed for the reference in Appendix D. We asked students to complete a similar 1D-element example for calculating stress/strain with different boundary conditions during a lab.

2.3 USE OF FEA SOFTWARE FOR STRESS/STRAIN ANALYSIS

The primary purpose of using SolidWorks simulation in lectures and labs is to demonstrate how to correctly use FEA software to run stress-strain analysis on components and assemblies under different boundary and loading conditions. The students are presented with a general procedure which can be applied to almost any commercially available FEA software.

The general procedure for FEA simulation in SolidWorks simulation with a concise description of each step is as follows:

(1) Pre-processing: The main tasks in this step are to remove unimportant features which have little effect on stress-strain analysis and to ensure 3D models can mesh.
(2) Define the type of analysis: SolidWorks simulation is an FEA simulation module of the SolidWorks platform and contains a variety of simulation types such as static analysis, thermal analysis, fatigue analysis, frequency analysis, linear dynamic analysis and more [19]. For MECH4200-Simulation-Based Design, the main focus was static analysis.
(3) Create a study: This creates a study name for simulation and all saved documents. Many different types of simulation could be run with different boundary conditions and element sizes.
(4) Define materials for each component: A proper material name with corresponding mechanical properties must be assigned to each component in order to run FEA simulation.
(5) Define connections or interfaces: For a single component's analysis, there is no need to specify the connections. For assemblies, however, contact relationships among components must be defined.
(6) Define restrains and loads: This is to specify loading and supporting conditions and explained as defining the boundary conditions.
(7) Meshing: In this step, by the proper settings for meshing such as meshing control and global meshing size, the 3D modeling will automatically mesh into thousands of small elements.
(8) Run the analysis: After settings are fully defined, the FEA software can solve the nodal deformations of all nodes. After the "run the analysis" is completed, all information about stress, deformation, and strain of the object is available.
(9) Post-processing: The main task of this step is to display stress, strain, and deformation of objects under consideration.
(10) Verification / Convergence: This step is to compare the FEA simulation results with theoretical calculation results and the results obtained from testing for the purpose of verification if available.

In teaching the MECH4200 Simulation Based Design course, we followed the above procedure to teach students how to appropriately use SolidWorks Simulation and to run the stress-strain analysis on both components and assemblies. Lectures expanded on the following topics: (1) Pre-process, (2) Type of elements, (3) FEA errors and convergence, (4) Meshing techniques, (5) Type of contacts and simplified theoretical connectors, (6) Loading and restraints, (7) Post-process and (8) Verification. During lectures and labs, detailed demonstrations of why and how to change every important setting were presented. Students then practiced and implemented what they learned through weekly case studies and two design projects

2.4 IMPLEMENTATION OF FEA SOFTWARE IN CLASS DESIGN PROJECTS

Conducting case study assignments was one way to let students practice and implement what they learned. Typically, each assignment was targeted to a specific skill or setting and most of the time the required setting for each homework was fully specified so that students could practice a specific skill and obtain the expected results. Conducting class design projects was another more effective way to practice and to implement what they learned to calculate stress-strain in an application. When conducting the class design projects, students were required to choose appropriate settings for the FEA simulation by themselves.

The course has two class design projects: a minor design project and a major design project. The individual minor project was designed to focus on skills for running an FEA simulation in a project-based environment. Students not only implemented necessary FEA simulation skills but also needed to make some decisions for a choice of setting. The ideal topic of a minor project should an issue which cannot be solved by a simple closed-form theoretical hand calculation. But there are the empirical formulas/curves or tables for providing acceptable solutions. The minor project was a 3-week individual project and was released in week 6 after lecturing and case studies for FEA simulation on components were completed. One minor project utilized the FEA simulation of stress concentration factors of components with various geometries and different loading for comparison to standard tables. Even though stress concentration factors cannot be obtained using a simple theoretical calculation, many handbooks and textbooks provide a set of curves to verify FEA results are within reasonable limits compared to theory.

The major project was a team-based project with 2~4 team members. It was released in week 9 after lecturing on FEA simulation including assemblies was completed. After the major project was released, one-hour lecture time per week was used to discuss the major project and to answer questions. The lecture was not to show how to run the FEA simulation of the major project but explained some possible approaches since a variety of potential approaches existed. One two-hour lab per week was devoted to working on the major project in a classroom while the instructor was available to provide guidance. Each design team was also required to spend at least an additional two hours outside of the classroom working on the major project. The team-based major design project was FEA simulation on a real product or a sub-assembly of a real product. A real product greatly increases students' interests in conducting FEA simulation and redesigning the product because the product requires real design constraints and specifications. The experience they gained through this project could be applied to almost any industrial hands-on real FEA simulation experience. One example of a major design project was the FEA simulation on an engine hoist, which is a common tool used in small workshops or homes to lift and move heavy objects.

3. DISCUSSIONS AND CONCLUSIONS

At the beginning of the course, students were reluctant to learn the fundamental concepts of finite element method theory. But they were forced to learn and study it because there was one exam covering this topic. After they learned fundamental concepts of FEM theory and manually conducting strain - strain analysis of bars under axial loading by using FEM theory, they seemed to appreciate learning fundamental concepts of FEM theory and had a better understanding of fundamental concepts of FEM theory. Based on direct observation, conversation, and students' feedback, the following were some perceived benefits of learning fundamental concepts of FEM theory.

- Students had a better understanding of the various types of elements because they went through several examples to build the approximate deformation functions of elements.

- They developed a better understanding of the errors encountered by the use of commercial FEA software. They knew that the FEA software would not provide exact accurate stress-strain results but provided approximate displacement results because the deformation functions of each element are only an approximate function.
- Students had a better understanding of meshing in the FEA software, which is a very important skill for applying FEA commercial software. They developed an understanding that the smaller size of the element would obtain much better results. They were able to apply the H-method and P-method for meshing in SolidWorks Simulation software [19]. The H-method is an automatic reduction of the size of elements to reach the convergence target. The P-method is an automatic increase in the order of the deformation functions while the size of the element is kept the same.
- Students knew that the FEA is actually to solve the deformations of each node or the deformation of each element. The stress/strain will be just a by-product, that is, stress and strain can be calculated after the deformation of each element are known.
- Learning the fundamental concepts of FEM theory and conducting manual stress-strain calculations by using the FEM theory significantly improved their understanding of applying the appropriate loading conditions and restraints.
- The manual stress-strain calculation of bars under axial loading by using FEM theory significantly improved their understanding of the procedure of applying FEA commercial software for stress-strain simulation of a component. They understood the purposes of key steps of the procedure.

In order to assess the effectiveness of teaching the fundamental concepts of FEM theory in the FEA course, we conducted a class survey during a lab at the end of the class. The survey data is presented in Table 1 and Table 2. Per Table 1, 92.3% of students agreed that providing lectures regarding fundamental concepts of the FEA method facilitated their understanding of the appropriate implementation of the FEA method in SolidWorks Simulation. Per Table 2, 92.3% of students agreed that continuing lectures of fundamental concepts of the FEA method should be kept as part of the course of MECH4200 Simulation Based design. The survey results were very positive and were in agreement with the finding in lots of previous publications [5, 6, 16, 17], that understanding some fundamental concepts of FEM theory would significantly improve students' ability to apply FEA software in engineering design.

Table 1 The results on the survey Question #1

Survey Question #1: "The lecturing about the fundamental concepts of the FEA method" facilitate me to understand the implementation of the FEA method in SolidWorks Simulation.					
Choices	Strongly agree	Agree	No opinion	Disagree	Strongly disagree
data	10	14	2	0	0

Table 2 The results on the survey Question #2

Survey Question #2: "The lecturing about the fundamental concepts of the FEA method" should be kept as part of the course MECH4200-Simulation-based design.					
Choices	Strongly agree	Agree	No opinion	Disagree	Strongly disagree
data	9	15	1	1	0

At the end of the MECH4200 Simulation Based Design course, students were asked to take the CSWA-S (The Certified SOLIDWORKS Associate – Simulation) certification exam. The CSWA-S certification exam is used to evaluate the students' understanding of the principles of the Finite Element Method (FEM) and their ability to use the SolidWorks Simulation to conduct the stress and strain simulation. The CSWA-S certification exam is hosted by the SolidWorks company. It is a two-and-half-hour certification test with a passing score of 70 out of 100 points. The administration fee for taking CSWA-S certification exam is $100. But our mechanical engineering program has 1000 education license and partners with the SolidWorks. Our students can take this exam with a waiver of this administration fee. In 2018 spring, there were a total of four sections of MECH4200 class with different instructors. We did not use the textbook for this course. However, we use the same syllabi to teach the same contents. In the author's section of the class, teaching the fundamental concepts of FEM theory was added as an additional method to help students to have a better understanding of fundamental concepts of FEA theory. The other sections were still a focus on the application of SolidWorks. The section with this approach of teaching the fundamental concepts of FEM theory had a pass rate of 60% (15 out of total 25 students) while the other four sections had an average pass rate of 35.9%, in which the focus of application of the SolidWorks Simulation was used. This result indicated that teaching the fundamental concepts of FEM theory facilitated students to use FEA commercial software.

Our experience with teaching FEA to undergraduate mechanical students indicated that teaching the fundamental concepts of FEM theory significantly helped them develop a better understanding of fundamental concepts of FEM theory and facilitated them to apply commercial FEA software for stress/strain simulations on components and assemblies.

4. REFERENCES

[1] Lee, R., Capece, V. R., and Baker, J., "*Integration Of Finite Element Software In Undergraduate Engineering Courses,"* 2001 ASEE Annual Conference and Exposition, June 16-19, Albuquerque, New Mexico.

[2] Zecher, J., "*Teaching Finite Element Analysis In An Met Program*," 2002 ASEE Annual Conference and Exposition, June 16-29, Montreal, Canada.

[3] Le, X., Moazed, A. R., and Duva, A. W., "*The Design Projects for the Simulation-Based Design Course,"* 2016 ASEE Annual Conference and Exposition, June 26-29, New Orleans, Louisiana.

[4] Papadopoulos, J. M., Papadopoulos, C., and Prantil, V. C., "*Philosophy of Integrating FEA Practice Throughout the Undergraduate CE/ME Curriculum,*" 2011 ASEE Annual Conference and Exposition, June 26-29, Vancouver, BC, 2011.

[5] Salamon, N., Wagle, G., and Lissenden, C., "*Applications Of Finite Element Analysis For Undergraduates,"* 2002 ASEE Annual Conference and Exposition, June 16-19, Montreal, Canada.

[6] Prantil, V., Musto, J., and Howard, W. (2001, June), "*Finite Element Analysis In A Mechanics Course Sequence,"* 2011 ASEE Annual Conference and Exposition, June 26-29, Vancouver, BC, 2011.

[7] Shih, R., "*The Applied Finite Element Analysis Course At Oregon Institute Of Technology,"* 2007 ASEE Annual Conference and Exposition, June 24-27, Honolulu, Hawaii.

[8] Pike, M., "*Introducing Finite Element Analysis In Statics,"* 2001 ASEE Annual Conference and Exposition, June 16-19, Albuquerque, New Mexico.
[9] Chaphalkar, P., and Blekhman, D., "*Introducing Finite Element Analysis In The First Course Of Statics And Solid Mechanics,"* 2007 ASEE Annual Conference and Exposition, June 24-27, Honolulu, Hawaii.
[10] Le, X., Olia, M., Moazed, A., and Roberts, R. L., "*Design a New Set of Strength Labs for the Course, 'Mechanics of Materials'," 2016* ASEE Annual Conference and Exposition, June 26-29, New Orleans, Louisiana.
[11] Steif, P., "*Use of Simplified FEA To Enhance Visualization In Mechanics Of Materials,"* 2004 ASEE Annual Conference and Exposition, June 20-23, Salt Lake City, Utah.
[12] Le, X., Duva, A. W., and Jackson, M., "*The Balance of Theory, Simulation, and Projects for Mechanical Component Design Course,"* 2014 ASEE Annual Conference and Exposition, Indianapolis, June 15-18, Indiana.
[13] Reffeor, W. S., "*Using FEA as a Pedagogical Tool for Teaching Machine Component Design,"* 2018 ASEE Annual Conference and Exposition, June 24-27, Salt Lake City, Utah.
[14] Ryan, R., "*Use of A Finite Element Stress Analysis Program To Enhance Learning In A Machine Design Course,"* 2004 ASEE Annual Conference and Exposition, June 20-23, Salt Lake City, Utah.
[15] Coyle, M., and Keel, C., "*Teaching Finite Element Analysis To Second Year Students,"* 2001 ASEE Annual Conference and Exposition, June 16-19, Albuquerque, New Mexico.
[16] N. Smith and J. Davis, *"Connecting Theory and Software: Experience with an Undergraduate Finite Element Course,"* 2015 ASEE Annual Conference and Exposition, June 14-17, Seattle
[17] Bearden, K., and Bandstra, J., "*Finite Element Analysis for The Mechanical Engineering Technology Student,"* 2001 ASEE Annual Conference and Exposition, June 16-19, Albuquerque, New Mexico.
[18] O.C. Zienkiewicz, R.L. Taylor, J.Z. Zhu, *"Finite element method: its basis and fundamentals,"* Amsterdam; Boston: Butterworth-Heinemann, 2013
[19] Paul Kuroski, *"Engineering Analysis with SOLIDWORKS Simulation 2017*", 2017, SDC publications

5. APPENDIX A: EXAMPLES FOR THE DISPLACEMENT FUNCTION

Example 1: Build the deformation function of a 1D element with two nodes and element length L_p as shown in Figure 2.

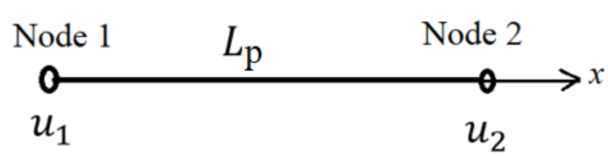

Figure 2 1D element with two nodes

Since this element only has two nodes, we will have two equations at these two nodal points where the deformation function has a value equal to the nodal deformation at the nodes. Therefore, because two equations can determine two unknowns, we can assume that deformation function is a linear function, as follows

$$u(x) = a_1 + a_2 x \tag{5}$$

Where a_1 and a_2 are two unknowns. At the two node points, we have:

At node 1, $x = 0$ and $u(x = 0) = u_1$, so $\quad a_1 + a_2 \times 0 = a_1 = u_1 \qquad (6)$

At node 2, $x = L_e$ and $u(x = L_e) = u_2$, so $\quad a_1 + a_2 \times L_e = a_1 + a_2 L_e = u_2 \qquad (7)$

From Equations (6) and (7), we have: $a_1 = u_1$ and $a_2 = (u_2 - u_1)/L_p$. Per Equation (5), the deformation function for this 1D 2 node element is:

$$u(x) = u_1 + \frac{(u_2 - u_1)}{L_p} x \qquad 0 \le x \le L_p \tag{8}$$

Per Equation (1) and Equation (8), we can determine the shape functions:

$$u(x) = N_1(x) u_1 + N_2(x) u_2 \tag{9}$$

$$N_1(x) = 1 - \frac{x}{L_p}, \qquad N_2(x) = \frac{x}{L_p} \tag{10}$$

Example 2: Build the deformation function of a 1D element with three nodes. In a local coordinate system, the coordinate values for node 1, node 2 and node 3 are -1, 0, and 1, respectively as shown in Figure 3.

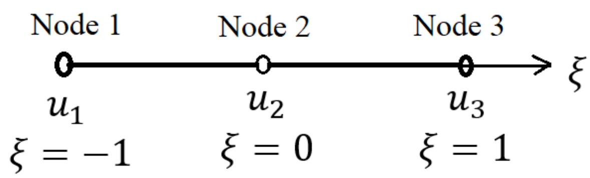

Figure 3 A 1D element with three nodes

Since this element has three nodes, the deformation function can be assumed to be a quadratic function: $u(\xi) = a_1 + a_2 \xi + a_3 \xi^2$. Per three equations at the node points and Per Equation (1), we can have the following deformation functions and the corresponding shape functions.

$$u(\xi) = u_2 + \left(\frac{u_3 - u_1}{2}\right) \xi + \left(\frac{u_1 - 2u_2 + u_3}{2}\right) \xi^2$$

$$u(\xi) = N_1(\xi) u_1 + N_2(\xi) u_2 + N_3(\xi) u_3$$

$$N_1(\xi) = -\frac{1}{2} \xi (1 - \xi); \quad N_2(\xi) = (1 - \xi)(1 + \xi); \quad N_3(\xi) = \frac{1}{2} \xi (1 + \xi)$$

Example 3: Build the deformation function and determine the corresponding shape functions of a 2D triangle element with three nodes as shown in Figure 4. The coordinates for the node 1, node 2 and node 3 are (0,1), (0,0) and (-1,0), respectively.

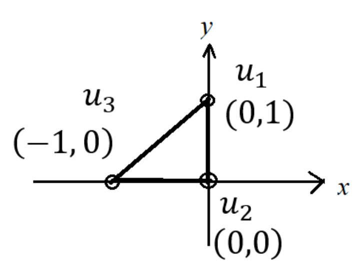

Figure 4 A triangle element with three nodes

Since the 2D triangle element has three nodes, the deformation function can be assumed to be a linear function: $u(x, y) = a_1 + a_2x + a_3y$. Per three equations at the node points and per Equation (1), we can have the following deformation functions and the corresponding shape functions.

$$u(x, y) = u_2 + (u_2 - u_3)x + (u_1 - u_2)y$$

$$u(x, y) = N_1(x, y)u_1 + N_2(x, y)u_2 + N_3(x, y)u_3$$

$$N_1(x, y) = y; \quad N_2(x, y) = 1 + x - y; \quad N_3(x, y) = -x$$

Example 4: Display the forms of deformation function for a tetrahedral element with four nodes and ten nodes. Derivation of deformation equations of a 3D element will be too tedious and is not necessary. However, it was worthy of mentioning that the deformation function for a 3D tetrahedral element with four nodes as shown in Figure 5 a) will be a linear function as:

$$u(x, y, z) = a_1 + a_2x + a_3y + a_4y$$

The deformation function for a 3D tetrahedral element with ten nodes as shown in Figure 5 b) will be a quadratic function as:

$$u(x, y, z) = a_1 + a_2x + a_3y + a_4y + a_5x^2 + a_6y^2 + a_7z^2 + a_8xy + a_9yz + a_{10}zx$$

a) tetrahedral element with four nodes   b) tetrahedral element with ten nodes

Figure 5 3D tetrahedral elements

6. APPENDIX B: AN EXAMPLE FOR ELEMENT PROPERTIES

Example 5: Establish element properties of a 1D element with two nodes as shown in Figure 6. The constant cross-section, length, and material Young's modulus of the 1D bar element are A_p, L_p and E_p, respectively. The two nodes are the node *i* and the node *j* with corresponding nodal deformation u_i and u_j, respectively. The internal forces on the nodes *i* and *j* are f_{ip} and f_{jp}, where the first subscript is the node number and the second subscript is the element number.

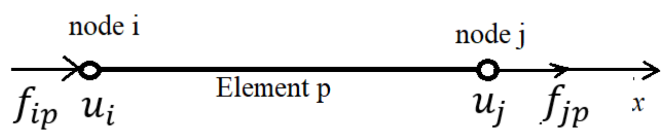

Figure 6 A general 1D element with two nodes

Per the result as shown in Equation (8), the deformation function of the element p will be:

$$u_p(x) = u_i + \frac{(u_j - u_i)}{L_p}x \qquad 0 \le x \le L_p \tag{11}$$

Using the Hooke's law, the stain and stress for this 1D element p will be:

$$\varepsilon_p(x) = \frac{du_p(x)}{dx} = \frac{(u_j - u_i)}{L_p} \qquad\qquad 0 \le x \le L_p \tag{12}$$

$$\sigma_p(x) = E_P \varepsilon_p(x) = \frac{E_P(u_j - u_i)}{L_p} \qquad\qquad 0 \le x \le L_p \tag{13}$$

From Equation (12) and (13), the 1D element with two nodes will have a constant strain and stress inside this element.

The potential energy Π_p of this 1D element P will be:

$$\Pi_p = \int_{V_p} \frac{1}{2} \sigma_P(x) \varepsilon_p(x) dV - \left(f_{ip} u_i + f_{jp} u_j\right) = \int_0^{L_p} \frac{1}{2} E_P \frac{(u_j - u_i)^2}{L_p^2} A_p dx - \left(f_{ip} u_i + f_{jp} u_j\right)$$

$$= \frac{1}{2} \frac{E_P A_p \left(u_j - u_i\right)^2}{L_p} - \left(f_{ip} u_i + f_{jp} u_j\right) \tag{14}$$

Using the minimum potential energy principle, we can use Equation (14) to get followings:

$$\frac{\partial \Pi_p}{\partial u_i} = 0 = \frac{E_P A_p \left(u_i - u_j\right)}{L_p} - f_{ip}, \quad \frac{\partial \Pi_p}{\partial u_j} = 0 = \frac{E_P A_p \left(u_j - u_i\right)}{L_p} - f_{jp}$$

Therefore, we have

$$\frac{E_P A_p}{L_p} \left(u_i - u_j\right) = f_{ip} \tag{15}$$

$$-\frac{E_P A_p}{L_p} \left(u_i - u_j\right) = f_{jp} \tag{16}$$

Equations (15) and (16) are called as element properties. They link together internal forces on the nodes f_{ip} and f_{jp}, the nodal deformation u_i and u_j, the material property E_P and the element geometrical properties A_p and L_p. Therefore, through the minimum potential energy principle, the stress/strain analysis problems are converted into a set of linear algebraic equations.

7. APPENDIX C: AN EXAMPLE FOR ASSEMBLY ELEMENTS

Example 6: Build the global linear algebraic equations of a problem with two 1D elements as shown in Figure 7. F_i represents external force on the node *i*. f_{ip} is the internal force on the node *i* of the element p. u_i is the nodal deformation of the node *i*. E_p, A_p and L_p are the Young's modulus, cross-section area and the length of the element *P*, respectively.

Use the results shown in Equations (15) and (16) in example 4, we obtain the element properties for element 1 and element 2. For element 1, we have:

$$\frac{E_1 A_1}{L_1} (u_1 - u_2) = f_{11} \tag{17}$$

$$-\frac{E_1 A_1}{L_1} (u_1 - u_2) = f_{21} \tag{18}$$

For the element 2, we have:

$$\frac{E_2A_2}{L_2}(u_2 - u_3) = f_{22} \tag{19}$$

$$-\frac{E_2A_2}{L_2}(u_2 - u_3) = f_{32} \tag{20}$$

Since only node 2 is shared, per loading conditions as shown in Figure 7, we have the following equations:

$$F_1 = f_{11}\,, \quad F_2 = f_{21} + f_{22}\,,\ F_3 = f_{32} \tag{21}$$

Using Equation (21) and Equations (17) ~ (20), we can get the global linear algebraic equations for two-1D-element system as followings:

$$\begin{gathered}
\frac{E_1A_1}{L_1}(u_1 - u_2) = f_{11} = F_1 \\
-\frac{E_1A_1}{L_1}(u_1 - u_2) + \frac{E_2A_2}{L_2}(u_2 - u_3) = f_{21} + f_{22} = F_2 \\
-\frac{E_2A_2}{L_2}(u_2 - u_3) = f_{32} = F_3
\end{gathered} \tag{22}$$

Equation (22) is the global linear algebraic equations. When loading conditions and restraints are given, we can use Equation (22) to solve nodal deformations, to fully define deformation functions of every element, and to further calculate strain and stress by using Hooke's law.

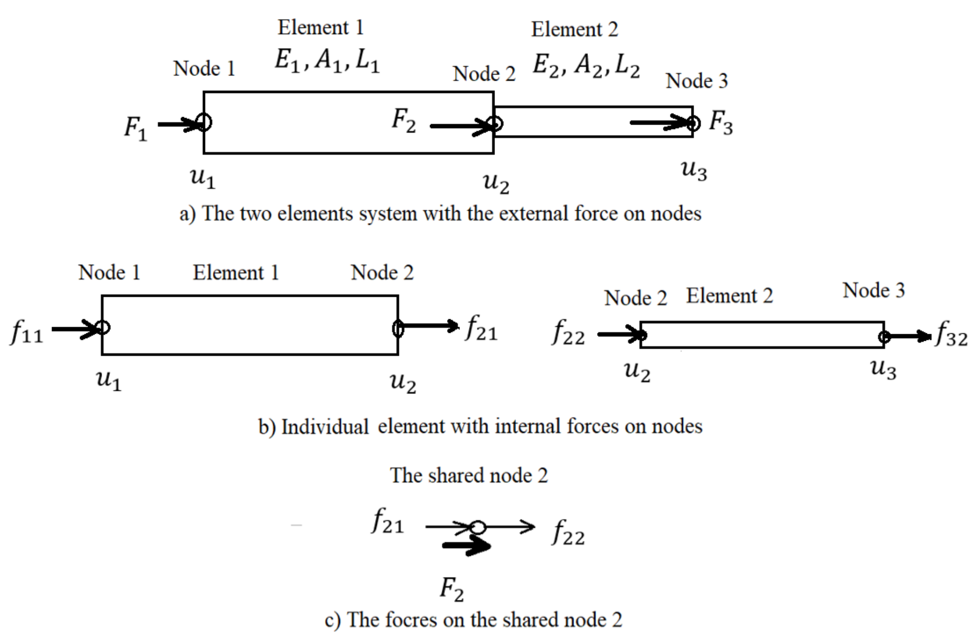

Figure 7 schematic views of a two-1D element system

8. APPENDIX D: A 1D EXAMPLE FOR CALCULATING STRESS/STRAIN

Example 7: An axial bar with two different constant-sections is under axial loading. A 100-N force is applied to the stepped cross-section as shown in Figure 8. Both ends of the bar are fixed.

The information for this bar is: $E = 210000(\frac{N}{mm^2})$, $F = 100\ (N)$, $L_1 = 100(mm)$, $L_2 = 100(mm)$, $\frac{EA_1}{L_1} = 3000(N.mm)$ and $\frac{EA_2}{L_2} = 1000(N.mm)$. Use the FEM theory to calculate strain, stress and reaction forces

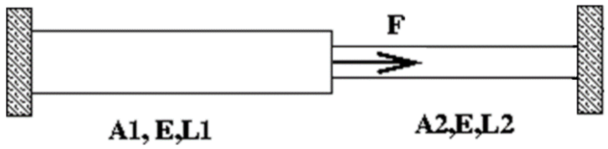

Figure 8 the schematic of a step-bar under axial loading

Step 1: Discretize the object.
The step-bar can be divided into two 1D elements. Each element will have a constant cross-section and two nodes. Figure 7a provides a free body diagram of the step-bar elements with the individual elements shown in Figure 7b.
Step 2: Deformation functions of each element
Since these 1D elements have only two nodes, the deformation function inside an element will be a linear function. Equation (8) of Example 1 can be directly used to get the deformation function of each element.
For element 1:

$$u_1(x) = \left(1 - \frac{x}{L_1}\right) u_1 + \frac{x}{L_1} u_2 \qquad 0 \le x \le L_1 \tag{a}$$

For element 2:

$$u_2(x) = \left(1 - \frac{x}{L_2}\right) u_2 + \frac{x}{L_2} u_3 \qquad 0 \le x \le L_2 \tag{b}$$

Step 3: Determine element properties by using the minimum potential energy principle
Equations (15) and (16) of Example 4 can be directly used to get the element properties.
For element 1

$$\begin{aligned} 3000(u_1 - u_2) &= f_{11} \\ -3000(u_1 - u_2) &= f_{21} \end{aligned} \tag{c}$$

For element 2

$$\begin{aligned} 1000(u_2 - u_3) &= f_{22} \\ -1000(u_2 - u_3) &= f_{32} \end{aligned} \tag{d}$$

Step 4: Assemble the global linear algebraic equations
The results of Example 5, that is, Equation (22) can be directly used to determine the global linear algebraic equations. Per Equation (22) and Equation (c) and (d), the global linear algebraic equations of the problem become:

$$\begin{aligned} 3000(u_1 - u_2) &= F_1 \\ -3000u_1 + 4000u_2 - 1000u_3 &= F_2 \\ -1000u_2 + 1000u_3 &= F_3 \end{aligned} \tag{e}$$

Step 5: Apply the boundary conditions
The loading and restrains, that is boundary conditions, for this example are:

$$F_2 = 100(N), \quad u_1 = 0, \quad u_3 = 0 \tag{f}$$

Apply the boundary conditions Equation (*f*) in Equation (*e*), we obtain the modified global linear algebraic equations for solving the problem.

$$-3000u_2 = F_1$$
$$4000u_2 = 100 \qquad (g)$$
$$-1000u_2 = F_3$$

Step 6: Solve the global linear algebraic equations

We can solve the Equation (*g*) to determine the nodal deformation and the reaction forces on the restraints as followings:

$$u_2 = 0.025(mm), \quad F_1 = -70(N), \quad F_3 = -25(N) \qquad (h)$$

Therefore, the nodal deformations of each node in this example are:

$$u_1 = 0, \quad u_2 = 0.025(mm), \quad u_3 = 0 \qquad (i)$$

Step 7: Calculate the strain and stress of the step-bar system

Using equations (*a*), (*b*) and (*i*), the deformation functions of element 1 and element are:

For element 1:

$$u_1(x) = 0.00025x \qquad 0 \le x \le 100(mm) \qquad (j)$$

For element 2:

$$u_2(x) = 0.025 - 0.00025x \qquad 0 \le x \le 100(mm) \qquad (k)$$

Using the definition of strain and the Hooke's law, we can calculate the strain and stress for the step-bar system.

Per Equation (*j*), the strain and stress of the element 1 will be:

$$\varepsilon_1(x) = \frac{\partial u_1(x)}{\partial x} = 0.00025$$
$$\sigma_1(x) = E\varepsilon_1(x) = 210000 \times 0.00025 = 52.5(MPa) \qquad 0 \le x \le 100(mm)$$

Per Equation (*k*), the strain and stress of the element 2 will be:

$$\varepsilon_2(x) = \frac{\partial u_2(x)}{\partial x} = -0.00025$$
$$\sigma_2(x) = E\varepsilon_2(x) = 210000 \times (-0.00025) = -52.5(MPa) \qquad 0 \le x \le 100(mm)$$