Hands-on Teaching in Finite Element Analysis to Undergraduate and Postgraduate Students

SAEID ZAHEDI VAHID, S. A. OSMAN & W. H. W. Badaruzzaman
Department of Civil & Structural Engineering
Faculty of Engineering & Built Environment
Universiti Kebangsaan Malaysia
43600 Bangi, Selangor
MALAYSIA
sdzahedi@yahoo.com ; saminah@eng.ukm.my

Abstract: - Finite element analysis (FEA) is an advanced numerical method of analysis in structural engineering. In most universities around the world, FEA is offered at undergraduate and postgraduate level due to vast amount of complex theoretical problems, which are beyond the capacity of students. This paper presents the observations and experiences on the use of general-purpose FEA software "LUSAS" in the teaching and learning of finite element method in structural problems of the undergraduate and postgraduate level by coursework at the Department of Civil and Structural Engineering, Universiti Kebangsaan Malaysia (UKM). A brief description on the theoretical course contents of FEA, strategic use of LUSAS software to supplement student's understanding of FEA theory and its use in the problem based FEA assignment is provided. It is observed that the use of LUSAS greatly helped students understanding of FEA structural applications. The feedback from students show that the exposure of LUSAS in the assignment project helps their development in the advanced analysis of structural engineering using finite element method and simultaneously increases their understanding of engineering practice for future study.

Key-Words: computer aided learning; finite element method; open node example; teaching strategies.

1 Introduction

Finite Element Analysis is probably the most popular numerical method used today. R. Courant first developed finite element analysis in 1943 [1]. He utilized the Ritz method of numerical analysis and minimized the calculus variation to obtain approximate solutions of vibration systems. It is also important to emphasize that the reliability of the FEA results is highly dependent on the skills and the basic theoretical knowledge of the engineer in the application of the method. Modern finite element developments have become very sophisticated, and the available software developed in the market nowadays has become very easy to use. It is important to ensure that the analyst, in his/her search for the best modeling method, correctly uses the tools available. It is more usual to teach FEA at the postgraduate level due to the complicated theories of the method. However, introducing FEA at the undergraduate level will create benefit to them as a basic preparation for further level [2]. To date LUSAS is known to be used as one of the popular choice in finite element method of analysis at universities and research institutes in Malaysia particularly in Faculty of Engineering and Built Environment, UKM. Through LUSAS, students will be provided with a number of illustrative uses of the software with the most recently added article at the top.

This paper presents the hands-on learning approach used to educate the civil engineering students in the fundamental of finite element theory and practices. The course outline offered to the students is also discussed, followed with a suggested procedure of developing the guidelines of FEA modeling in structural engineering problems.

2 Teaching and learning approach in FEA

Previously studying in FEA only dealt with pages of mathematical equations before students being exposed to any meaningful examples. In some other cases, the education and exposure of the subject was involved nothing more than the presentation of a slide show without hands-on classes were allowed. Based on the authors' observation, students always face difficulties in doing their project with finite element software because of the vague guideline. Although there are many options of FEA software in the market nowadays and each of it has its own applications and limitations, knowing the software

ISBN: 978-1-61804-155-5

abilities will helps student in completing the coursework.

In the effort to avoid the said problems, a step by step guidelines have been developed in which would benefit students in using FEA. The developed guidelines are as follows:-

- Use LUSAS or any commercial package
- Use the determined case study, which is compulsory and randomly given to the student.
  - Get students to work in groups to develop a team-working approach
  - Concentrate upon the development of a detailed guideline for each case study by the students.
- Investigate the parameters which can determine an appropriate balance between accuracy, efficiency and cost-effectiveness.
  - Preparing a manual for all case studies that has been done and presented by the student.

3 Finite Element Course and Educational Objectives

The course syllabus that normally offered to the students will cover an introduction to the finite element method and fundamental concept; one dimensional problems; trusses; two-dimensional problems using constant strain triangles; computer applications and project.

At the end of the semester, the course aims at producing student who is able to acquire the basic knowledge and understanding of the finite element method. The ability of students to apply the FEA knowledge for solving any kind of engineering problems that specifically related to structure is also expected. Student ability to run commercial finite element package such as LUSAS will also be one of the important outcomes of the course. Along with the above, student must be able to demonstrate their ability to be an effective team member with leadership quality and would be able to acquire new information related to the finite element method via lifelong learning [3].

During the course, hands-on class will be provided in two hours for introducing and training of LUSAS which involves guiding them through a simple elastic example that illustrates many of the package's features.

4 Hands-on project

The course in FEA requires students to carry out a group assignment project through LUSAS. The assignment involved with modeling of one of the case studies as shown in Table 1. Students were asked to solve the problem based on the following main headings:

- Description contains a summary of the example, defining geometry, material properties, analysis requirements and results processing requirements.
- Objectives state the aims of the analysis.
- Keywords contain a list of keywords as an aid to selecting the correct examples to run.
- Modeling contains procedures for defining the features and attribute datasets to prepare the LUSAS model file. Multiple model files are created in some of the more complex examples and these contain more than one 'Modeling' section.
- Running the Analysis contains details for running the analysis and assistance should the analysis fails for any reason.
- Viewing the Results contains procedures for results processing using various methods.

Table 1. List of Case Studies

FEM	Static Analysis	Linear	-Buckling -Load Carry Capacity -Stress-Strain
		Non-Linear	-Load Control -Displacement Control
	Dynamic Analysis	Linear	-Modal Analysis -Eigen Value Analysis -Seismic Analysis -Moving Load
		Non-Linear	-Implicit (low cycle) -Explicit (High Cycle)

To solve the given case studies, students were advised to refer to LUSAS Manual that will be provided throughout the semester. The manual contains useful general information about the Modeler User Interface and details of how the examples are formatted.

The first example in this manual contains detailed information to guide you through the procedures involved in developing the LUSAS model, running the analysis and viewing the results. The example details out the contents of each dialog, the necessary text entry and mouse commands involved. The remaining examples will assume that students have completed the fully worked example and may not necessarily contain the same level of information.

The examples are of varying complexity and cover different modeling and analysis procedures using LUSAS. It will benefit students to work through as many examples and commands as possible. Most of the examples given are written for use with the base versions of LUSAS V14 software products. The LUSAS software product and any product options that are required will be stated in the beginning of the example.

Except where mentioned, all examples are written to allow modeling and analysis that need to be carried out with the teaching and training version of LUSAS which has restrictions on the problem size. The limitations size of each problem is currently set as given in Table 2.

Table 2. Allowable model size in teaching and training version

500 Nodes	100 Points	250 Elements	1500 Degrees of Freedom	10 Load cases

The best way to illustrate the simulation modeling example that actually used based on the application of these principles is as given below:

A nonlinear plane stress analysis is to be carried out on a model of a reinforced concrete continuous beam. The beam tested by Mansur [4] was chosen and need to be analyzed through FEA. The reinforcement is provided in the lower face of the beam and has a total cross-sectional area of 400 mm^2. The superposition of nodal degrees of freedom assumes that the concrete and reinforcement are perfectly bonded. It is assumed that the self-weight of the beam is negligible compared with the applied load and the effects of any shear reinforcement can be ignored.

A concentrated vertical load is applied on the top of the beam 1200mm from the left-hand support. The concrete section is meshes by plane stress (QPM8) elements, and the reinforcement bars are meshed by bar (BAR3) elements. A nonlinear concrete cracking material model will be applied to the plane stress elements and a von Misses plastic material will be applied to the reinforcement bars. The project details are shown in Fig.1.

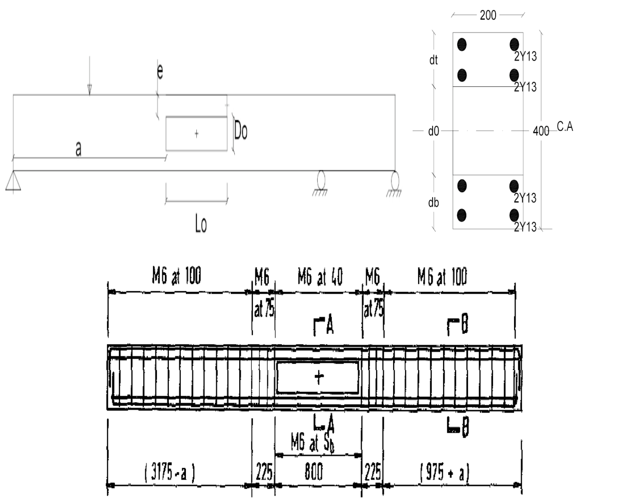

Fig.1. Project Details

Through FEA students were asked to investigate the behavior of the beam under cracking/yielding condition and the results must be produced based on the following requirements:

- A Deformed Mesh Plot showing the final deformed shape.
- A Load Displacement Graph for the top node on the axis of symmetry of the beam.
- Stress contour plot showing the stress distribution in the beam.
- Crack pattern plot showing the crack patterns produced.
- Animation of stresses and crack patterns for selected load increments.
- A graph of variation in stress through selected slice sections through the beam.

To ensure that the modeling in FEA will represent the actual behavior of the reinforced concrete continuous beam, a verification study is needed in this analysis. Students need to compare their LUSAS results with the tested values. In this simulation, the ultimate load obtained from the test was 135 kN, whilst the ultimate load obtained from LUSAS was 148 kN. Therefore, the difference is less than 10 % and acceptable. Fig.2 shows the comparison study on maximum load and the maximum deflection value of the beam from LUSAS with that of the laboratory test results. It was proven that LUSAS is reliable in predicting the behaviour of concrete beam.

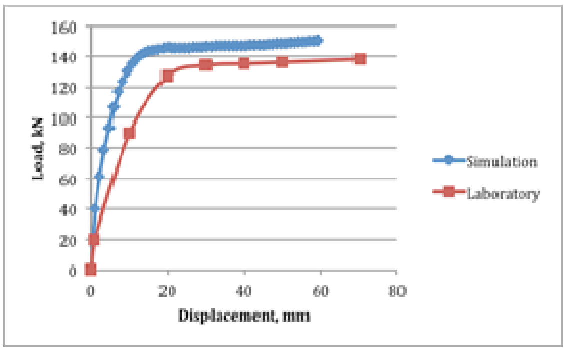

Fig.2. Load versus Maximum Deflection Curves

Finally after completing the simulation modeling, the report for the nonlinear finite element analysis of reinforced concrete continuous beam with openings must include a step by step procedure as listed below:

- Creating a new model
- Defining the Geometry
- Defining Groups
- Defining the Mesh-Reinforcement Bars
- Defining the Mesh-Concrete
- Defining the Geometric Properties
- Defining the Material Properties
- Assigning Attributes to the Bars
- Assigning Attributes to the Concrete
- Making all groups visible
- Supports
- Loading
- Nonlinear Control
- Running the Analysis
- Viewing the Results
- Deformed Shape
- Creating a Load versus Displacement Graph using the Graph Wizard
- Principal Stress Contour Plots
- Viewing Crack Patterns
- Animating the Results
- Creating a slice section of results

The project report submitted by the group will be one of the examples of the prepared manual that can be used for final year students in preparing their final thesis. The reports may show them how to model the different structural problem with variety of boundary conditions and the result that involved in each of case studies.

5 Assessment and students' feedback

The assessment of the course can be divided into two parts in which the first part (70%) will assess the student basic knowledge and understanding of the course throughout the semester. These can be achieved based on direct assessment from the tutorial, mid semester and final examination. Whilst in the second part of the assessment (30%), the psychomotor or generic skills of the students in using LUSAS software were assessed based on their presentation and report.

In order to observe the effectiveness of the teaching and learning approach through hands-on class, student's feedback towards the course were gathered. Students were asked to provide their opinion at the end of the semester. Most of their feedbacks were very positive in regards of the way the FEA was taught. The learning approach via hands-on class has provided them with great opportunities and simultaneously increases their understanding of the FEA basic in solving any structural problems. Some of the students' comments on the course are:

- The FEA tutorials were very helpful.
- Becoming more knowledgeable in LUSAS.

6 Conclusions

As a conclusion, by introducing hands-on class has greatly helped students in learning and understanding on the important of FEA in structural engineering applications. Students' feedback at the end of semester showed that they enjoyed the learning process and simultaneously developed a better appreciation of the potential afforded by FEA. The submitted report of the FEA project can be used as an example or a guideline for the final year students in preparing their dissertation especially for those who have selected simulation study as their interest.

Acknowledgements

The authors wish to acknowledge the financial support received from the Centre for Engineering Education Research, Universiti Kebangsaan Malaysia as research grant (PTS-2011-017) in the effort of improving the quality of teaching and learning in engineering education.

References:

[1] Giuseppe Pelosi, *The finite-element method, Part I: R. L. Courant: Historical Corner*, Antennas and Propagation Magazine, Vol, 49, No 2, 2007, pp. 180-182.

[2] Zhuge, Y. & Mills, J.E., Teaching finite element modelling at the undergraduate level: a PBL approach, *Proceedings of Australian association of engineering education*, Adelaide, 2009, pp. 1-6.

[3] Self Assessment Report Engineering Accreditation Council (EAC), Department of Civil & Structural Engineering, Faculty of Engineering & Built Environment, UKM, 2010

[4] Mansur, M.A., Lee,Y.F., Tan, K.H., & Lee, S.L., Tests on RC Continuous Beams with Openings, *Journal of Structural Engineering*, ASCE, Vol, 117, No. 6, 1991, pp. 1593- 1606.