APPLICATION O   ı FINITE ELEMENT MODELLING OF TWO DIMENSIONAL TRUSS

HASIF BIN ROSAZANAM

A thesis submitted in partial fulfillment of the requirements for the award of the degree of Bachelor of Civil Engineering

Faculty of Civil Engineering & Earth Resources

Universiti Malaysia Pahang

MAY 2011

ABSTRACT

Finite element analysis (FEA) has become common place in recent years, and is now the basis of a multibillion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important. This study is about modeling of Two dimensional truss structure using software Ansys 12.0 (APDL) and determines the maximum deflection, maximum stress and maximum strain when the load is applied to the model. Ansys is software based on the finite element analysis (FEA) widely used in the computer-aided engineering (CAE) field. Four random inputs will be used in this study which is Young Modulus (E), Poison Ratio, Radius (R) and Fy loads. The random output will be the graph maximum of deflection, stress and strain. All of the output values should be passing the eight equations of statistics to make sure the values are valid. This study is going to be basic guidelines on how to implement the civil engineering element using Ansys software.

ABSTRAK

Analisis unsur terhingga (FEA) telah menjadi perkara biasa sejak beberapa tahun kebelakangan ini, dan sekarang menjadi asas kepada berbilion nilai industri setahun. Penyelesaian numerikal walaupun untuk masalah stress yang sangat complex sekarang ini dapat diselesaikan dengan mudah menggunakan FEA menjadikan kaedah ini sangat penting. Kajian ini merangkumi kaedah pemodelan rangka bumbung menggunakan software Ansys 12.0 (APDL) dan menentukan nilai maksimum defleksi, nilai maksimum stress dan nilai maksimum strain apabila beban dikenakan. Ansys adalah software berasaskan FEA yang digunakan meluas di dalam bidang kejuruteraan perkomputeran. Empat data rawak akan dimasukkan iaitu Young Modulus (E), Poison Ratio, Radius (R) dan beban menegak. Semua data yang dihasilkan perlu melepasi 8 persamaan statistik untuk memastikan data itu boleh diguna pakai. Kajian ini akan menjadi rujukan asas untuk menerapkan unsur kejuruteraan awam menggunakan Ansys software.

TABLE OF CONTENT

CHAPTER	TITLE	PAGES
	TITLE	i
	DECLARATION	ii
	DEDICATION	iii
	ACKNOWLEDGEMNET	iv
	ABSTRACT	v
	ABSTRAK	vi
	TABLE OF CONTENT	vii
	LIST OF TABLE	xii
	LIST OF FIGURE	xiii
1	**INTRODUCTION**	
	1.1 Preliminaries	1
	1.2 Problem Statement	3
	1.3 Objective of Study	3
	1.4 Scope of Study	4

2 LITERATURE REVIEW

2.1 Finite Element Analysis 5

2.2 Finite Element Principal Steps 7

2.3 *History of Finite Element Method* 8

2.4 Finite Element Formulation 9

2.5 Matrix analysis of trusses 9

2.6 Previous research that relate to the finite element of the truss 10

2.6.1 Nonlinear positional formulation for space truss analysis 10

2.6.2 Finite element model updating of a truss model using incomplete modal data 11

2.6.3 FEM subsystem replacement techniques for strength problems in variable geometry trusses 12

2.6.4 Nonlinear finite element modeling and characterization of guyed towers under severe loading 13

2.6.5 Finite element analysis of double angle truss connections 13

2.6.6 The ANSYS Probabilistic Design System 14

2.6.7 Design of truss-structures for minimum weight using genetic algorithms 16

2.6.8 Geometrically Nonlinear Arc Length Sparse Finite Element Analysis and Optimal Design of Truss Structures 17

2.6.9 *Logarithmic strain measure applied to the nonlinear positional formulation for space truss analysis* 18

2.6.10 Strain and stress relation for non-linear finite element simulations 19

2.6.11 Non-linear closed-form computational model of cable trusses 20

2.6.12 Geometrically nonlinear shell finite element based on the geometry of a planar curve 21

2.7 Parametric Design Analysis for Evaluating a Range of Variables 21

2.8 Benefits of Parametric Design Analysis 22

2.9 Previous research that relate to the APDL 23

2.9.1 Investigation on determination of loading path to enhance formability in tube hydro forming process using APDL 23

2.9.2 Probabilistic finite element analysis using ANSYS 24

2.9.3 Finite element limit load analysis of thin-walled structures by ANSYS (implicit), LS-DYNA (explicit) and in combination 24

3 **METHODOLOGY**

3.1 Introduction to Ansys 26

3.2 Steps in Ansys 26

3.3 Modelling of the Two Dimensional Truss Model 30

3.3.1 Give the simplified version a title 31
3.3.2 Enter key points 32
3.3.3 Form Lines 34
3.3.4 Defining the Type of Element 35
3.3.5 Define Geometric Properties 36
3.3.6 Element Material Properties 38
3.3.7 Mesh Size 39
3.3.8 Mesh 39
3.4 Solution Phase Assigning Loads and Solving 40
3.4.1 Define Analysis Type 40
3.4.2 Apply Constraints 41
3.4.3 Apply loads 42
3.4.4 Solving the System 43

4 RESULT AND DISCUSSION

4.1 Introduction 44
4.1.1 Deformation of Model 45
4.1.2 Nodal Solution 45
4.2 Random Input Data 46
4.2.1 Graph Distribution of Height 47
4.2.2 Graph Distribution of Width 48
4.2.3 Graph Distribution of Young Modulus 49
4.2.4 Graph Distribution of Load 1 50
4.2.5 Graph Distribution of Load 2 51
4.2.6 Graph Distribution of Load 3 52
4.2.7 Graph Distribution of Load 4 53
4.3 Random Output Data 54
4.3.1 Graph of Maximum Deflection 54
4.3.2 Graph of Maximum Strain 55
4.3.3 Graph of Maximum Stress 56

4.4 *Eight Equations of Statistics* 57

4.4.1 Error Sum of Squares 57

4.4.2 Coefficient of Determination 57

4.4.3 Maximum Relative Residual 58

4.4.4 Constant Variance Test 58

5 CONCLUSION AND RECOMMENDATION

5.1 Conclusion 62

5.2 Recommendation 62

REFERENCES 63

LIST OF TABLE

NO.TABLE	TITLE	PAGES
3.1	Keypoints to Modeling the 2D Truss	32
4.1	Random Input Variables	46
4.2	Goodness-of-Fit Measures for Random Output Parameter Maximum Deflection	59
4.3	Goodness-of- FitMeasures for Random Output Parameter Maxvonmisesstress	60
4.4	Goodness-of-Fit Measures for Random Output Parameter Maxvonmisesstrain	61

LIST OF FIGURE

NO.FIGURE	TITLE	PAGES
2.1	First finite element mesh for the analysis of dam gravity	8
2.2	Probabilistic Design System integrated into ANSYS	15
3.1	Stage Diagram of Ansys	26
3.2	Main Window	28
3.3	Output Window	29
3.4	Model of the Two Dimensional Truss Structure	30
3.5	Change Title	31
3.7	Create Keypoints	33
3.8	Insert Keypoints	33
3.9	Form Lines	34
3.10	The completed line elements	35
3.11	Adding Element Type	36
3.12	Element Type	36
3.13	Define Geometric Properties	37
3.14	Adding Geometric Properties	37
3.15	Element Material Properties	38
3.16	Adding Element Material Properties	38
3.17	Mesh Size	39
3.18	The completed mesh without node numbers.	40
3.19	Define Analysis Type	41
3.20	Apply Constraints	42
3.21	Apply Loads	42
3.22	Completed 2d Truss Structure	43

4.1	*Geometry Model*	44
4.2	Deformation of Model	45
4.3	Nodal Solution	45
4.4	Graph Distribution of Variable Height	47
4.5	Graph Distribution of Variable Width	48
4.6	Graph Distribution of Young Modulus	49
4.7	Graph Distribution of Load 1	50
4.8	Graph Distribution of Load 2	51
4.9	Graph Distribution of Load 3	52
4.10	Graph Distribution of Load 4	53
4.11	Graph of Maximum Deflection	54
4.12	Graph of Maximum Strain	55
4.13	Graph of Maximum Stress	56

CHAPTER 1

INTRODUCTION

1.1 Preliminaries

Civil engineers design and construct major structures and facilities that are essential in our everyday lives. Civil engineering is perhaps the broadest of the engineering fields, for it deals with the creation, improvement and protection of the environment, providing facilities for living, industry and transportation, including large buildings, roads, bridges, canals, railroad lines, airports, water-supply systems, dams, irrigation, harbors, docks, tunnels, and other engineered constructions. Over the course of history , civil engineers have made significant contributions and improvements to the environment and the world we live in today.

The work of a civil engineer requires a lot of precision. This is mainly due to the fact that the final result of any project will directly or indirectly affect people's lives; hence safety becomes a critical issue. Designing structures and developing new facilities may take up to several months to complete. The volumes of work, as well as the seriousness of the issues considered in project planning, contribute to the amount of time required to complete the development of an adequate, safe and efficient design.

The introduction of software usage in the civil engineering industry has greatly reduced the complexities of different aspects in the analysis and design of projects , as well as reducing the amount of time necessary to complete the designs. Concurrently, this leads to greater savings and reductions in costs. More complex projects that were almost impossible to work out several years ago are now easily solved with the use of computers. In order to stay at the pinnacle of any industry, one needs to keep at par with the latest technological advancements which accelerate work timeframes and accuracy without decreasing the reliability and efficiency of the results.

Some common Structural Analysis & Design Software available in the market:

• STADD III:

Comprehensive structural software that addresses all aspects of structural engineering- model development, analysis, design, visualization and verification.

• AXIS VM: (http://www.axisvm.com)

Structural analysis and design with an updateable database of element sections and specifications available in the market.

• ANSYS: (http://www.ansys.com)

All-inclusive engineering software dealing with structural analysis and other engineering disciplines such as fluid dynamics, electronics and magnetism and heat transfer

• ETABS:

Offers a sophisticated 3-D analysis and design for multistory building structures.

1.2 Problem Statement

In Malaysia, the main problem for a lot of engineering firms is the lack of exposure to computer analysis software that is used by modern countries such as USA , United Kingdom , Sweden and etc. Furthermore this research will benefit future upcoming companies looking to save cost rather than wasting millions of dollars handling test in construction labs.

Other than that, our country has reached a stagnant level in all major economic sectors especially in the construction industry. The fact that our country doesn't implement on technology builds the burden on design engineers and consultants workscope and productivity. Very few Universities in Malaysia have had the opportunity to fully utilize and use computer analysis programs for either research use or common study for their students.

Therefore, this research is handled to expose and utilize the usage of Ansys software to make calculations of designing structures more productive and efficient.

1.3 Objectives of the Study

The objectives of the study is:

i. To study data of finite element of beams using Ansys software
ii. To identify the models used such as finite element beams and implementing different test and analysis methods onto a particular beam model.
iii. To give a chance for future young engineers to develop the skill to obtain knowledge using computer analysis software , in this case using Ansys software.
iv. To fully increase productivity of engineers using computer analysis programs rather than using hand-calculations

1.4 Scope of Study

The location of this study done in UMP Pekan campus using the campus mechanical laboratory. Research has been done on finite element models through past journals on beams. Prior to that, our supervisor Dr. Cheng Hock Tian M. Sc, B. Eng.(Civil) has trained and teached us on operating Ansys software. Other than that we also joined Ansys programming courses to further increase our understanding on using the software. All data of Ansys software collected will be studied further in order to know the total quantity and composition of the beam model studied. From the data and knowledge obtained, potential usage of this software can be used later in the working field.

CHAPTER 2

LITERATURE REVIEW

2.1 Finite Element Analysis

Finite element analysis (FEA) has become commonplace in recent years, and is now the basis of a multibillion dollar per year industry. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important.

Finite element codes are less complicated than many of the word processing and spreadsheet packages found on modern microcomputers. Nevertheless, they are complex enough that most users do not find it effective to program their own code.

A number of prewritten commercial codes are available, representing a broad price range and compatible with machines from microcomputers to supercomputers. However, users with specialized needs should not necessarily shy away from code development, and may find the code sources available in such texts as that by Zienkiewicz2 to be a useful starting point. Most finite element software is written in Fortran, but some newer codes such as felt are in C or other more modern programming languages. (David Roylance , 2001)

Finite Element Method is a numerical approach in solving the problem arising in a physics and engineering. This method gives approximate solutions to differential equation that model the problems. Basically, the finite element method requires a problem defined in geometrical space to be subdivided into a finite number or smaller regions. (Pepper, D.W and Heinrich J.C, 2006)

The beginning of the finite element was due to the frustration in the attempt to use different method on more difficult geometrical irregular problems. The early use of finite elements lay in the application of such techniques for structurally related problems. (Pepper, D.W and Heinrich J.C, 2006)

By dividing the into a large number of smaller part of elements and using appropriate compatibility and equilibrium equations to assemble these elements, it is possible to obtain an almost accurate prediction of values of variables such as stress and displacement of body. Consequently, the smaller the elements are divided, the more accurate the solution is but a cost of increased computation time.

The main features of FEM are:

- The entire solution domain is divided into small finite segments.
- Over each element, the behaviour is described by displacement of the elements and the material law.
- All elements are assembled together and the requirements of continuity and equilibrium are satisfied between neighbouring elements.
- The solution matrix is lightly polluted
- FEM is very suitable for practical engineering problems of complex geometric. To obtain the good accuracy in the region of rapidly changing variables, a large number of small elements need to be used.
  (Pepper, D.W and Heinrich J.C, 2006)

2.2 Finite element principal steps.

In practice, a finite element analysis usually consists of three principal steps:

Preprocessing: The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or "elements," connected at discrete points called "nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical "preprocessor" to assist in this rather tedious chore. Some of these pre-processors can overlay a mesh on a pre-existing CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process

Analysis: The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.

Postprocessing: In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. A typical postprocessor display overlays coloured contours representing stress levels on the model, showing a full-field picture similar to that of photo elastic or moiré experimental results. (C.A. Brebbia 1982, Zienkiewicz and R.L. Taylor, 1989)

2.3 History of Finite Element Method

Significant finite element research was conducted at the University of California at Berkeley during the period 1957 to 1970. The initial research was a direct extension of classical methods of structural analysis which previously had been restricted to one-dimensional elements. The majority of the research conducted was motivated by the need to solve practical problems in Aerospace, Mechanical and Civil Engineering.

During this short period the finite element method was extended to the solution of linear and nonlinear problems associated with creep, incremental construction or excavation, crack closing, heat transfer, flow of water in porous media, soil consolidation, dynamic response analysis and computer assisted learning of structural analysis. During the last six years of this period the fields of structural analysis and continuum mechanics were unified.

The computer programs developed during this early period at Berkeley were freely distributed worldwide allowing practicing engineers to solve many new problems in structural mechanics. Hence, the research was rapidly transferred to the engineering profession. In many cases the research was used professionally prior to the publication of a formal paper. (Ray W. Clough & Edward L. Wilson, 1999)

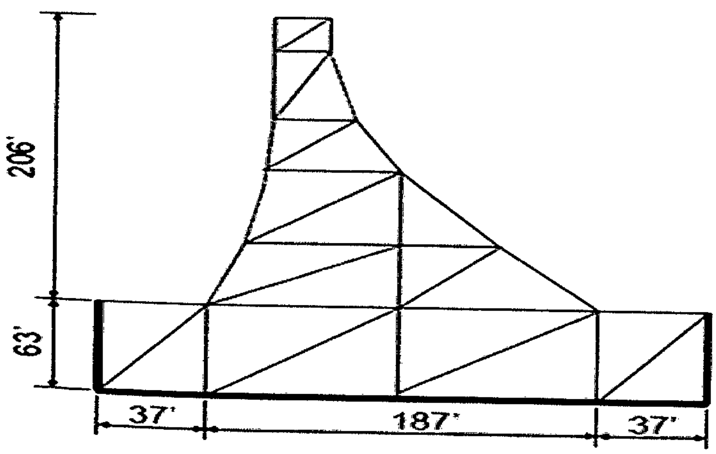

Figure 2.1: First finite element mesh for the analysis of dam gravity

(Ray W. Clough & Edward L. Wilson,1999)

2.4 Finite Element Formulation

The finite element description is developed for planar mechanisms with revolute joints. Making an assumption that the material of column elements is linearly elastic (follows Hooke's law) with uniform mass density p and uniform cross-sectional area A, the element stiffness matrix [k], of the ith element is in the same form as for a planar truss element (C. S. Desai, Elementary Finite Element Method. Prentice-Hall, Englewood Cliffs, NJ, 1979).

2.5 Matrix analysis of trusses

Pin-jointed trusses provide a good way to introduce FEA concepts. The static analysis of trusses can be carried out exactly, and the equations of even complicated trusses can be assembled in a matrix form amenable to numerical solution. This approach, sometimes called "matrix analysis," provided the foundation of early FEA development. (David Roylance , 2001)

Matrix analysis of trusses operates by considering the stiffness of each truss element one at a time, and then using these stiffness's to determine the forces that are set up in the truss elements by the displacements of the joints, usually called \nodes" infinite element analysis. Then noting that the sum of the forces contributed by each element to a node must equal the force that is externally applied to that node, we can assemble a sequence of linear algebraic equations in which the nodal displacements are the unknowns and the applied nodal forces are known quantities. (David Roylance , 2001)

Either the force externally applied or the displacement is known at the outset for each node, and it is impossible to specify simultaneously both an arbitrary displacement and a force on a given node. These prescribed nodal forces and displacements are the boundary conditions of the problem. It is the task of analysis to determine the forces that accompany the imposed displacements and the displacements at the nodes where known external forces are applied. (David Roylance , 2001)

2.6 Previous research that relate to the finite element of the truss

2.6.1 Nonlinear positional formulation for space truss analysis

The structural element known as the space truss is widely employed in Structural Engineering, particularly in designs involving large spans. Numerical modeling of space structures involves nonlinearity generated by geometrical changes that occur in the structure and nonlinearity generated by the behaviour and instability of materials. (M Greco, Gesualdo, 2005)

The research presents a new method based on the finite element method to solve static elastoplastic problems with large deflections. The proposed formulation, which complements the formulation published by Coda and Greco 2004, uses a simple engineering strain measure.

The method exhibits a high degree of convergence and accuracy, and the number of iterations decrease as the number of degrees of freedom increase. The formulation can analyze severe geometrical nonlinear behaviour, including structural post-buckling behaviour. The four numerical examples presented yielded highly accurate responses compared with analytical and other numerical solutions.

The formulation can be extended easily to the three-dimensional modelling of solids, which would simply require doing integrations in volume elements in the three main directions of stress. In that case, three stress components would be considered in the energy function instead of one, as in the current formulation, and transformations of coordinates would be required in the elements

Based on the finite element method (FEM), the proposed formulation uses nodal positions rather than nodal displacements to describe the problem. The strain is determined directly from the proposed position concept, using a Cartesian coordinate system fixed in space. Bilinear constitutive hardening relations are considered here to model the elastoplastic effects, but any other constitutive model can be used. (M Greco, Gesualdo, Venturini, Coda, 2004)

2.6.2 Finite element model updating of a truss model using incomplete modal data

The experimental updating results show that the model updating approach greatly improves the finite element model such that the analytical modal parameters better match the measured modal parameters. Research presented the application of a two-step finite element model updating methodology to a three-dimensional truss bridge model. (Y.X. Zhang and S.H. Sim & B.F. Spencer, Jr)

First, the sensitivity of the change in modal parameters to variations in the physical parameters was analyzed. Based on these sensitivity analyses, the parameters that can be updated were determined. Next, the optimization of an orthogonally based objective functional was employed in model updating using the modal data obtained from measurements on a physical structure.

Sequential Quadratic Programming (SQP), a quasi-Newton approach that creates a linear search strategy applicable to non-linear minimization problems was used to acquire a precise and stable solution. In computing the solution, the gradients of every parameter versus the objective function were determined by executing quadratic fitting methods at each iterative step in order to improve the solution's precision. The results of the experimental updating show that the analytical model is improved after model updating such that the natural frequencies from the finite element model better match the experimentally determined natural frequencies. (Y.X. Zhang and S.H. Sim & B.F. Spencer, Jr)

2.6.3 FEM subsystem replacement techniques for strength problems in variable geometry trusses

Variable geometry structures are those capable of modifying their geometry to adapt to different loads and working conditions. This is possible because some of the elements comprising them can vary their length. These elements are called actuators. Another characteristic making them interesting is their high stiffness to weight ratio, which has contributed to the application of variable geometry structures in the spatial research field. The study of these structures dates back to the 1980s. (K. Miura, H. Furuya, D. Gokhale, 1985)

The study cantered on replacing the finite element model of an articulated variable geometry truss (VGT) joint (with an elevated number of 3D elements including contact elements) with an equivalent parametric microelement (EPM) with few elements. (Luis M. Macareno, Josu Agirrebeitia, Carlos Angulo, RafaelAvilés, 2007)

Therefore, a method was defined to minimize an objective function based on the equivalence of elastic strain energy absorbed by the two models. Thus, the aim is to find EPM parameter values which minimized this function. The optimization method is based on the resolution of a nonlinear redundant equation system, tackled via the nonlinear least squares method. For this an iterative diagram was proposed where the equations are linearized via the first order Taylor series development and subsequently solved via a least squares method. This optimization is based on an initial solution via genetic algorithms. (Luis M. Macareno, Josu Agirrebeitia, Carlos Angulo, RafaelAvilés, 2007)

2.6.4 Nonlinear finite element modelling and Characterization of guyed towers under severe loading

Nonlinearity in structural analysis comes from three common sources: geometric nonlinearity, material nonlinearity, and boundary condition nonlinearity. Since this research will focus on nonlinear behaviours of towers under severe loading, a series of reviews have been conducted. Research about geometric nonlinearity in the structural members (truss, cable and beam) will be presented first. General material nonlinearity will be introduced afterwards. The algorithms that implement static analysis and dynamic analysis will be briefly reviewed as well. Finally, the evaluation method for beams under impulsive loading will be introduced. (Haijian Shi, 2007)

Literature (Bhatti, 2005; Madenci, 2006; Zienkiewicz, 2005) has explored the formulations of basic elements in FEM. But most of them only discussed about linear elements, which do not adjust element stiffness matrix according to deformation. Because in a large deformation scenario the influence of changing geometry is considerable and cannot be neglected, the fully nonlinear elements developed by (Pai, 2007) are introduced herein.

2.6.5 Finite element analysis of double angle truss connections

Shear lag can be described as a phenomenon that creates a loss in resistance in a tension member connected through only part of its cross-section. It is a complex problem which has been under study for many years by researchers. Parameters that influence the shear lag phenomenon are many and difficult to assess: type and size of cross-section, type of connection, length of welds, length of member, joint eccentricities, etc. Connections between double-angle web members and chords in trusses or open web steel joist are considered herein. (A. Benaddi, D. Bauer, 2003)