ADVANCED COMPUTER MODELLING TECHNIQUE FOR FINITE ELEMENT ANALYSIS OF TRUSS STRUCTURE

MOHD AB HAKIM BIN IBRAHIM

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

DECEMBER 2010

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 planar 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

LIST OF TABLE

LIST OF FIGURE

CHAPTER 1

INTRODUCTION

1.1 General

Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centred on the "stiffness and deflection of complex structures".

By the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Since the rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision. Present day supercomputers are now able to produce accurate results for all kinds of parameters.

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction.

Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

There are generally two types of analysis that are used in industry: 2-D modelling, and 3-D modelling. While 2-D modelling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modelling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively.

Within each of these modelling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

Nowadays, there are a lot of software that can calculate the finite element method. For example ABAQUS, ACTRAN, ALGOR, FEM-Design and ANSYS.

For this project, we use the Ansys software to simulate the finite element of structure. ANSYS, Inc. is an engineering simulation software provider founded by software engineer John Swanson. It develops general-purpose finite element analysis and computational fluid dynamics software.

While ANSYS has developed a range of computer-aided engineering (CAE) products, it is perhaps best known for its ANSYS Mechanical and ANSYS Multiphysics products.

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces called elements. The software implements equations that govern the behaviour of these elements and solves them all creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behaviour of the product, be it electromagnetic, thermal, mechanical etc.

Structural analysis is probably the most common application of the finite element method as it implies bridges and buildings, naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

- Static Analysis - Used to determine displacements, stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

- Transient Dynamic Analysis - Used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.

- Buckling Analysis - Used to calculate the buckling loads and determine the buckling mode shape. Both linear (eigenvalue) buckling and nonlinear buckling analyses are possible.

In addition to the above analysis types, several special-purpose features are available such as Fracture mechanics, Composite material analysis, Fatigue, and both p-Method and Beam analyses.

1.2 Problem Statement

In this research, model of planar truss should be create using Ansys software. After that, Ansys software will be simulated the program to take the data of deflection, stress and strain. This data need to be determined to identify the limit of planar truss structure before failed due to load applied. This study is another alternative way on how to determine the finite element analysis using software.

a) How to run the finite element analysis of truss using Ansys software
b) How to implement the civil element to the Ansys software

1.3 Objective

a) The objective of this research is to determine the deflection, stress and strain value on planar trusses when the load is applied using finite element method by Ansys software.
b) To test the equation that get from the software using the eight equation of statistic.

1.4 Scope of work

The scope of work is consisting of software exploration. We need to be expert in Ansys software to make sure we are able to create the model and do the analysis. To learn and get used to the software, a lot of exploration need to do. For the start, we are learning from University of Alberta tutorial for simple model. Besides that, we also should study more on model like truss structure. The study consists of truss behaviour and how load will affect the structure generally and more specific, how load affect to the deflection, stress and strain of truss structure.
What we need to finish this project are:

a) Modelling the planar truss
b) Simulate the model using Ansys software.
c) Get the data of deflection and stress strain of the model

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)

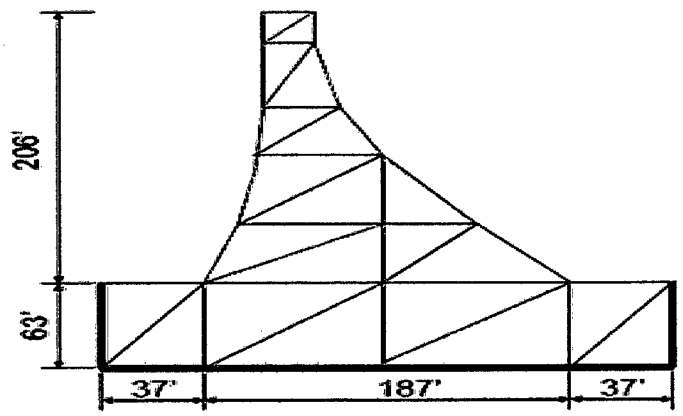

Figure 2.1: First finite element mesh for the analysis of dam gravity (Ray W. Clough & Edward L. Wilson,1999)

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)