ANALYSIS OPTIMIZATION USING NUMERICAL EVALUATION METHODS FOR 2D TRUSS STRUCTURE

NURUL ASHIKIN BT. ABDULL RAHMAN

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

Faculty of Civil Engineering and Earth Resources
University Malaysia Pahang

NOVEMBER 2010

ABSTRACT

The study of the 2D truss structure can be done in laboratory by conducting laboratory test. But, usually these tests are expensive and time consuming. With the advance of computer technology and development of finite element software packages, the finite element method may provide an alternative way to analyze the 2D truss structure. In this study, the 2D truss transmission tower analyzed by using finite element ANSYS. In this structure, there are two sources of the data which are come from input parameter and output parameter graph. Output parameters are consists of maximum deflection, Maximum Stress and Maximum Strain while input parameter consists of real constant, horizontal and vertical force, Poison's ratio, and Young's Modulus. ANSYS software has advantages of factorial design and be able to generate analysis in graph without change the range and run it again. Several goodness-of-fit measures are provided for each random output parameter.

ABSTRAK

Kajian mengenai kekuda yg mempunyai struktur 2 dimensi boleh dijalankan di makmal dengan melakukan beberapa ujikaji.Tetapi, pada kebiasaanya,ujikaji tersebut adalah sangat mahal untuk dijalankan dan memakan masa. Dengan kemajuan dalam teknologi perkomputeran dan pembangunan dalam pakej perisian unsur terhingga,kaedah unsur terhingga mungkin sesuai sebagai cara alternatif untuk mengkaji struktur kekuda dua dimensi. Oleh itu,dalam kajian, struktur kekuda 2 dimensi dianalisis menggunakan perisian unsur terhingga ANSYS. Dalam struktur ini, terdapat dua jenis data iaitu graf parameter dalaman dan parameter luaran. Parameter luaran termasuk nilai maksimum pemesongan, nilai maksimum tegangan dan nilai maksimum tegasan manakala parameter dalaman termasuklah real constant, daya melintang dan menegak, nisbah Poison's, dan Modulus Young. Perisian ANSYS mempunyai banyak kebaikan dalam reka bentuk pemfaktoran kerana berupaya menganalisis graf tanpa menukar nilai dalam lingkungan dan menganalisis semula. Beberapa nilai yang ditetapkan dapat mengukur parameter luaran sama ada menepati dalam pengiraan atau tidak.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 General

ANSYS is general-purpose finite element analysis (FEA) software. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces called elements. 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.

There are several generic steps to solving problem in ANSYS, first of all build the geometry (construct a two or three dimensional representation), define material properties of the structure, generate mesh, apply loads on the structure, obtain solution and present the results. In general finite element solution may be separate into the following three stages. This is the guideline used for setting up any finite which is pre-processing is used for defining the problem and determine the major steps of analysis.

Besides, solution stage divided into three parts, assigning loads, constraints and solves the resulting of the equations. The last stage is post processing which is further processing and viewing of results.

1.2 Problem Statement

In the real situation without calculation by software, it might be difficulties to the engineer to calculate all the calculation start from the foundation until the roof. The calculations are consists of the long calculation in order to make sure the design is safe and economical. Besides, the long manually calculations can cause a lot of mistakes and the alternative way is using the ANSYS software to avoid the mistakes. In this case, ANSYS is used to analyse 2D truss structure (power transmission tower) which consists of point load. According to the ANSYS parameter the deflection, stress and strain will be defined in the range that we set up. It will be easy when just put the range and the program will process according to the range without build the model or change the dimension. The graph of the parameter which is deflection, stress and strain will get in the final result by using APDL (Advanced Parametric Design Language).

1.3 Objectives

The objectives in analyse the 2D truss structure by using the ANSYS are:

i. To analyze a power transmission tower using ANSYS software.
ii. To determine deflection at each joint.
iii. To determine reaction forces at the base.
iv. To determine the stress in each member, strain and the deflection at graph and the result of the truss structure.

1.4 Scope of Study

To achieve the objectives, scopes have been identified in this research. The scopes of this research are analyzing the design of a power transmission tower using ANSYS. Model of truss structure are designed using finite element analysis. Moreover, the ANSYS program must be studied by completes all the tutorials from University Alberta which is related to the topic.

APDL is using to calculate the parameters which the parameters must success and pass. The APDL is ANSYS parametric design languages scripting language that automate common tasks or even build model in terms of parameters. ANSYS commands can be used as part of the scripting language and APDL commands discussed here are the true scripting commands and encompass a wide range of other features. Complete sets of elements behavior, material models and equation solvers for a wide range of engineering problems.

A power transmission tower is a common example of a structure that is made up of only truss members. These towers are actually 3-D structures, but for the sake of simplicity it will take a cross-sectional face of the tower. The tower is mainly subjected to loading in the vertical direction due to the weight of the cables. Also it is subjected to forces due to wind. In this example we will consider loading due to the weight of the cables, which is in the vertical and horizontal direction.

The model of the transmission tower is build to analyze the critical deflection of the structure, stress and strain of the structure. After the model is done using the ANSYS software, put the range of the input variables. Input variables for this model are RX1 (real constant), FX1 (horizontal direction), FX2 (horizontal direction), FY1 (vertical direction), PRXY (poison's ratio), and EX (Young's Modulus). Besides the output parameters for this model are maximum deflection, maximum Von Misess Stress, and maximum Von Misess Strain.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Over the last few years electronic data processing has changed the work of engineers throughout all the fields of construction. This is particularly true for design of structures where, nowadays it is unimaginable can be done without the help of the computer software. Even simple structure like, for example simply supported reinforced concrete beams under uniform loading, are designed using the help now available from computers. In many cases, these computer calculations are faster, less costly and thus more profitable than manual calculations. With the increasing complexity of a numerical model, it becomes more likely that important details are overlooked, due to a lot of the information produced by computer. The finite element calculations are illustrated, not just by theoretical systems, but also by the relating to real structures.

2.2 ANSYS Analysis

ANSYS is a general purpose finite element method the numerical that consists of solutions to a lot of problems in engineering. The finite element method (FEM) is the most popular simulation method to predict the physical behavior of systems and structures. Since analytical solutions are in general not available for most daily problems in engineering sciences numerical methods have been evolved to find a solution for the governing equations of the individual problem. *(*S. Moaveni,1999*)*

The finite element analysis (FEA) method, originally introduced by Turner *et. al* (1956), is a powerful computational technique for approximate solutions to a variety of "real world" engineering problems having complex domains subjected to general boundary conditions. In effect Finite Element Analysis reduces the problem to that of a finite number of unknowns by dividing the domain into a elements and by expressing the unknown field variables in term of the assumed approximating functions within each element.

Moreover in ANSYS analysis, there are three main steps in a typical analysis which are model generation. Model generation are consists of simplifications, idealizations, define materials and material properties and generate finite element model(mesh). Second step is solution, it consists two elements such as specify the boundary conditions and obtain the solution. The third step is review results, which are plot or lists the results and check for validity to the analysis. .(Madenci and Guven,2006).

ANSYS is one of the most influential finite element analysis software in the world because of its very powerful calculation and analysis ability. ANSYS software is widely used in finite element analysis, but its preprocess function is so complex for complex model. (Rui and Jianmin, 2008)

2.3 ANSYS History

The finite element method is a numerical procedure that can applied to obtain solutions to a variety of problems in engineering. The origin of the modern finite element method may be traced back to early 1900s when some investigators approximated and modeled elastic continue using discrete equivalent elastics bar. ANSYS is a comprehensive general-purpose finite element computer program that contains over 100,000 lines of code. ANSYS is a very powerful and impressive engineering tool that may n]be used to solve a variety of problems.(Saaed Moaveni,1999)

The finite-element method originated from the need for solving complex elasticity and structural analysis problems in civil and aeronautical engineering. Its development can be traced back to the work by Alexander Hrennikoff (1941) and Richard Courant (1942). While the approaches used by these pioneers are dramatically different, they share one essential characteristic: mesh discretization of a continuous domain into a set of discrete sub-domains, usually called elements. (Giuseppe Pelosi,2007)

Hrennikoff's work discretizes the domain by using a lattice analogy while Courant's approach divides the domain into finite triangular subregions for solution of second order elliptic partial differential equations (PDEs) that arise from the problem of torsion of a cylinder. Courant's contribution was evolutionary, drawing on a large body of earlier results for PDEs developed by Rayleigh, Ritz, and Galerkin.

Development of the finite element method began in earnest in the middle to late 1950s for airframe and structural analysis and gathered momentum at the University of Stuttgart through the work of John Argyris and at Berkeley through the work of Ray W. Clough in the 1960s for use in civil engineering. By late 1950s, the key concepts of stiffness matrix and element assembly existed essentially in the form used today. (Weaver and Gere ,1966)

The method was provided with a rigorous mathematical foundation in 1973 with the publication of Strang and Fix's *An Analysis of The Finite Element Method.* (Gilbert and George 1973)

2.4 ANSYS Parametric Design Language

APDL stands for ANSYS Parametric Design Language, a scripting language that can be use to automate common tasks or even build your model in terms of parameters (variables). APDL also encompasses a wide range of other features such as repeating a command, macros, if-then-else branching, do-loops, and scalar, vector and matrix operations.

While APDL is the foundation for sophisticated features such as design optimization and adaptive meshing, it also offers many conveniences that can be use in our day-to-day analyses.

ANSYS uses two types of parameters which are scalar and array. The parameter can be use instead of a literal number as an argument to any ANSYS command. The parameter is evaluated and its current value is used for that argument. If the parameter name is input in a numeric argument of a command, the numeric value of the parameter is substituted into the command at that point. The parameters names must be begin with a letter, contain only letters, numbers, and underscore characters and the parameters are contain no more than eight characters. (Brzev and Pao,2006)

When using APDL to make a parametric model, it is necessary to avoid directly using numbers of various objects for operating. Whereas, it's better to use the selecting functions, such as position selecting, association selecting, subordination selecting and property selecting to achieve parametric modeling. (Haghpanahi and Mapar, 2006)

APDL, (Ansys Parametric Design Language) was used to write FEA programs which can automatically build the model, solve it ,retrieve the results and do other complicated operations and calculations. By using this kind of programs, FEA becomes a powerful tool to do DOE (Design of Experiment) and design optimizations. (Lam Tim Fai, 1997)

2.5 Design Optimization

Optimization means minimization or maximization. There are two broad types of design, a functional design and an optimized design. A functional design is one that meets all the preestablished design requirements, but allow for improvements to be made in certain areas of design. Design optimization is always based on some criterion such as cost, strength, size, weight, reliability, noise or performance.(Saaed Moaveni,1999)

2.6 Probabilistic Design System

As the name implies, the ANSYS Probabilistic Design System (PDS) is made to address probabilistic problems. As such, it can be used for an uncertainty analysis or a reliability analysis. It is tightly integrated into ANSYS, using the same graphical user interface, hence having the same look and feel as ANSYS itself. The targeted audience is advanced engineers who are comfortable with performing finite element analyses using ANSYS on a regular basis. The PDS is based on the ANSYS Parametric Design Language (APDL), which allows users to parametrically build a finite element model, solve it, obtain results and extract characteristic results parameters such as the maximum stress for example. In addition, APDL provides the possibility to use input variables as well as result parameters in arithmetic expressions, do-loops and if-then-else constructs. (Reh, Beley, *et. al* ,2006)

2.7 Probabilistic analysis

The underlying principle of the method is that the input parameters of the model are defined, not by a single value, but by a statistical distribution. This distribution can take any form that can be defined mathematically. A common example is the Normal, or Gaussian, distribution that can be uniquely defined by a mean and standard deviation.

A probabilistic analysis can be implemented in a variety of ways. The simplest is to select random numbers and use these to generate random values from the distribution. A variety of ways exist for sampling a distribution in this way, some of which are described by Comack an Shuter(1991). More efficient methods, requiring fewer model solutions are available (Reh,2000).

2.8 Deterministic Analysis.

A deterministic analysis is the transformation function representing the relationship between the input variables influencing the behaviour of a product and the result parameters characterizing the product behaviour. In simple cases the result parameters can be expressed as an analytical function, but in realistic cases the input-output relationship is only given algorithmically for example using finite element program. (Reh, Beley *et. al* ,2006)

2.9 Factorial Design

The early part of the twentieth century, scientific experimentation largely followed the "vary one variable at a time" method. Fisher (1925) developed factorial experimental designs and it was Taguchi (1987) that introduced these methods into the design process itself. Factorial design enables the estimation of the sensitivity of a system to variation in a large number of input parameters whilst reducing experimental effort.The Taguchi method lends itself well to FEA. The terminology used is familiar to engineers, and the methodology relies more on engineering judgment than absolute statistical values. In structural FEA, it is usually required to determine one or more response variables, such as maximum von Mises stress, minimum nodal displacement and so on.

2.10 Response surface method

Response surface methods avoid the disadvantages of Monte-Carlo simulation methods by replacing the true input–output relationship by an approximation function.

2.10.1 Regression Analysis

The traditional regression analysis is usually applied to homogeneous observations. However, there are several real situations where the observations are not homogeneous. In these cases, by utilizing the traditional regression, we have a loss of performance in fitting terms. Then, for improving the goodness of fit, it is more suitable to apply the so-called clusterwise regression analysis.(D'Urso and Santoro,2006).

2.10.2 Accuracy and validity

Response surface methods are based on the assumption that the response surface is an adequate representation of the true input–output relationship. It is also assumed that the residuals are normally distributed with constant variance. The goodness-of-fit measures implemented in both ANSYS tools to check the validity.

2.10.3 Monte-Carlo simulation methods

The key functionality of Monte-Carlo simulation techniques is the generation of random numbers with a uniform distribution from 0 to 1. The PDS uses the L'Ecuyer algorithm for generating these random numbers, while the DesignXplorer uses the much more advanced Mersenne Twister algorithm. (Mastumoto and Nishimura,1998)

The interpretation of the results of a Monte-Carlo simulation analysis is based on statistical methods. The statistical procedures to calculate for example mean values, standard deviations and to derive histogram plots are available in textbooks on statistics and probabilistic methods (Ang and Tang,1975)

2.10.4 Advantages of Monte-Carlo simulation methods

The Monte-Carlo simulation method does not make any simplification or assumptions in the deterministic or probabilistic model. The only assumption it does in fact make is that the limited number of samples is representative to quantify the randomness of the result parameters. The error associated with this assumption is well quantifiable as outlined above. With increasing the number of samples the simulation of Monte-Carlo simulation method converges to the true and correct probabilistic result. The Monte-Carlo simulation is therefore widely used the benchmark to verify the accuracy of other probabilistic methods. Another advantage is the fact that the required number of simulations is not a function of the number of input variables. (Reh, Beley, *et. al* ,2006)

2.11 Batch files

Commands that gives general access to the ANSYS processor always star with slash(/). For example /PREP command gives general access to the ANSYS preprocessor. To leave a processor and return to the Begin Level, issue the FINISH command. The fundamental tool used to enter data and control the ANSYS program is the command. Some commands can be used only at certain places in your batch file, while others may be used in other processors. The command format consists of one or more fields separated by commas. The first fields always contain the command name. A command argument may be skipped by not specifying any data between the commas. In such cases, ANSYS substitutes the default data for the argument.(Saaed Moaveni,1999)

2.12 Static Analysis

A truss that is assumed to comprise members that are connected by means of pin joints, and which is supported at both ends by means of hinged joints or rollers, is described as being statically determinate. Newton's Laws apply to the structure as a whole, as well as to each node or joint.

The behavior of structures under static loading can be analyzed by employing different types of elements within ANSYS. The nature of the structure dictates the type of elements utilized in the analysis. Under certain types of loading and geometric conditions, the three-dimensional type of analysis can be idealized as two dimensional analysis (Madenci and Guven,2006).

2.12.1 Trusses

In architecture and structural engineering, a truss is a structure comprising one or more triangular units constructed with straight members whose ends are connected at joints referred to as nodes. External forces and reactions to those forces are considered to act only at the nodes.

In engineering truss structure consisting of straight members connected at their ends by means of bolts, rivets, pins or welding. Trusses offer practical solutions to many structural problems in engineering, such as power transmission tower, bridges and roofs of buildings. (Saaed Moaveni,1999)

A truss is a structure that is made of straight structural members capable of carrying loads only in their own direction, i.e. no shear forces, no moments. Thus, each member is under either axial tension or axial compression. These members are connected to each other by means of joints. It is assumed that loads can only be applied at joints. (Madenci and Guven,2006).

Truss is an assemblage of straight members connected at their ends by flexible connections to form a rigid configuration. Because of their light weight and high strength, trusses are widely used, and their applications range from supporting bridges, transmission tower, and roofs of the building to being support structures in space stations.(Kassimali,2005).

2.12.2 Basic Truss Element

The stable internally stable (or rigid) plane truss can be formed by connecting three members at their ends by hinges to form a triangle. This triangular truss is called the basic truss element. This triangular truss is internally stable in the sense that it is a rigid body that will not change its shape under loads. (Kassimali,2005)

A truss is composed of triangles because of the structural stability of that shape and design. A triangle is the simplest geometric figure that will not change shape when the lengths of the sides are fixed. (Ricker and Clifford ,1912)