DEVELOPMENT OF TRUSS FINITE ELEMENT ANALYSIS SOFTWARE USING ANSYS

NURUL FARHANA BTE MOHAMAD

A report submitted in partial fulfillment of the
Requirement for the award of the degree of
Bachelor of Civil Engineering

FACULTY OF CIVIL ENGINEERING AND EARTH RESOURCES
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2010

ABSTRACT

Truss is one of the important components for a structure and need to be designed in such a way that they have enough strength and rigidity to satisfy the strength and serviceability limitation. Although there are many existing commercial engineering software such as STAAD.Pro, LUSAS, MATLAB and etc, but most of them are expensive and it is hard for those small companies to get the license easily. Finite Element analysis has been reported to be suitable tool to analysis the truss. This is because finite element analysis is cheaper and easier to conduct compared to full scale test. Here, the study was conducted to prove finite element analysis using nonlinear analysis is able to give a similar result as the lab test. This report presents the development of truss finite element analysis software using ANSYS. The purpose of this study is to develop a cheaper home-grown truss analysis software using ANYS. A step-by-step Finite Element formulation of simple truss is discussed and Graphical User Interfaces (GUIs) was developed so as to provide user-friendly environments. At the end of the study, results generated from the truss analysis software are compared with the existing engineering software STAAD.Pro. for validation.

ABSTRAK

Kekuda merupakan salah satu daripada bahagian utama dalam sesuatu struktur dan perlu direkabentuk supaya mencapai kekuatan dan kekukuhan yang mencukupi untuk memenuhi had kekuatan dan had kebolehkhidmatan. Walaupun, terdapat banyak perisian yang diciptakan dalam pasaran misalnya STAAD.Pro, LUSAS dan MATLAB, tetapi kebanyakannya harganya yang tinggi menyebabkan agak sukar bagi syarikat-syarikat kecil untuk mendapatkan lesen dengan senang. Analisis kaedah unsur terhingga dikatakan merupakan suatu cara yang sesuai untuk analisis kekuda. Ini kerana analisis kaedah unsur terhingga adalah lebih mudrah dan senang untuk dijalankan berbanding dengan ujian skala penuh. Kajian ini adalah bertujuan untuk membuktikan analisis kaedah unsur terhingga menggunakan analisis ketidaklelurusan adalah mampu memberikan keputusan yang hampir sama dengan ujian makmal. Laporan ini menerangkan tentang pembanguan perisian untuk menjalankan analisis element unsur terhingga terhadap kekuda dengan menggunakan ANSYS. Objektif kajian ini adalah membangunkan perisian analisis element unsur terhingga terhadap kekuda yang lebih murah dengan menggunakan ANSYS. Langkah-langkah formulasi element unsur terhingga telah dibincangkan dan Graphical User Interfaces (GUIs) telah digunakan untuk lebih memudahkan kepada para pengguna. Akhirnya, keputusan yang diperolehi dari perisian analisis kekuda dibandingkan dengan keputusan dari STAAD.Pro untuk membuktikan kebolehannya.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 General

Truss is very important for a construction, such as construction for roof, bridge and high rise building. Truss can give high esthetic value for mega construction such as Eiffel Tower, Paris and for building like stadium for football in Europe. 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 and result in forces in the members which are either tensile or compressive forces. Moments are explicitly excluded because all the joints in a truss are treated as revolute.

Nowadays, the analysis of truss is concerned of many designers and consultants. The truss structures are required to be designed in such a way that they have enough strength and rigidity to satisfy the strength and serviceability limitation. In order to archive the minimum requirement, it is necessary to carry out an accurate analysis to investigate the reaction and stress that acting inside the member of the truss. When the load acting on a truss, the structure may deform and change to different shape or size.

This can be a result of compression (pulling) stresses or tension (pushing) stresses inside the truss members. For addition information, a compression will occurs when there is a result of the subjection of a material to compression force that induces the reduction of volume. Besides, a tension will occurs when there is a pulling force with a magnitude subjected to an object and results an elongation. Furthermore, it can be deformed or moved from its original position after the load is applied. So, there will be a displacement (deflection) occur at nodes. In order to come out with a proper design of truss, it is necessary for all the designers and consultants to examine all these important data and include them into their design.

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 element. The software implements equations that govern the behavior 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 is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering as well as the Physics and Chemistry departments.

ANSYS provide 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 behavior of the product, be it electromagnetic, thermal, and mechanical and many more.

APDL is stand for ANSYS parametric design language, a scripting language that automate common tasks or even build model in terms of parameters. While all ANSYS commands can be used as part of the scripting language, the APDL commands discussed here are the true scripting commands and encompass a wide range of other features. ANSYS structural offers a comprehensive product solution for structural linear or non linear and dynamics analysis. Complete sets of elements behavior, material models and equation solvers for a wide range of engineering problems.

1.2 Problem Statement

In most of the cases, a structure will fail and collapse due to the improper design of the structural components. Among of them, the unfit design of truss is one of the factors that cause a structure in unstable and unsafe for the users. However, people are always concentrating on the design of main components of a building structure (such as column, beam, slab and etc.) compare to the design of truss component. Some of them are just annoy the truss design and replace it with a common design standards used in the construction. But, sometimes the design standards used are unable to fulfill the requirement all the time. As an engineer, it is necessary for them to calculate and design the truss then carry out a most suitable truss design.

Normally, the design of truss can be carried out easily by using the existing engineering software in the market. The consultant or designer can just fill in the material information (such as length, Young' Modulus, area and etc) into the software, model the structure to generate the results. But in some cases, the consultants or designers may produce inaccurate results because some of them do not have enough knowledge on the related software. The inaccurate of data from analysis may bring the truss unstable and cannot function well due to insufficient of strength.

Engineers need to know the deflection, strength, and stress and strain for a structure before designing it. However they have to calculate the same structure for a couple of time. A lot of time has been waste by do so. But, by using ANSYS we can put range for the pressure so that the same structure can be designed with different loads. This will help the engineers to make their work easier because they do not need to repeat the same calculation.

1.3 Objectives

Every thesis has its own objectives. The objective is depends on what we want to achieve. Objective is goal for everything that we want to do. This thesis has a few objectives that related to the structure and analysis. So many things that we need to find to realized these objectives. The objectives are:

i) To obtain the deflection parameter for truss.

ii) To find out the parameters that get from the software using the eight equation of statistic.

iii) To simplify the method that used by the engineer.

1.4 Scope of Works

Work that I need to do is learn how to use the ANSYS software. I learn it by using the tutorial in the internet. Before use the ANSYS software I must really know how to use it. Besides the ANSYS software, I also have to learn about Advanced Parametric of Design Language (APDL). This programmed is the advance version of ANSYS. Then, I must run the model to get the equation using this software.

I also need to really understand about truss. How many type that commonly used in construction, and I must know about this. Truss has a few purposes that I must really understand. I must know how to design a truss.

CHAPTER 2

LITERATURE REVIEW

2.1 ANSYS and Finite Element Analysis

ANSYS is the most comprehensive and advanced finite element based civil or structural engineering simulation code proven through years of use. It combines the leading engineering simulation software, ANSYS, and the high-end civil engineering specific structural analysis capabilities. ANSYS Structural software addresses the unique concerns of pure structural simulations without the need for a lot of extra tools. These are general-purpose finite element modeling packages for numerically solving mechanical problems, including static or dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

ANSYS is a major product for computer based Prototyping. While material Prototyping is as old as mankind, the computerized approach is relatively new and has on offer certain major advantages over the classical approach. In the last decades a new method came to maturity, which adds more flexibility, cost efficiency and, especially, more insight into prototyping. This more recent method is known under names like FEA (finite element analysis), simulation or virtual (digital) prototyping, and moves the material prototype, the experimental verification, into the computer. Simulation is a great step ahead, allows precise prognosis and optimization of the performance of parts and products, with all the flexibility of a computerized model which can change dimensions, materials, loads without the necessity to create a new material prototype. The range of physics problems that can be analyzed is basically unlimited which can be it mechanical or thermal loads, be it a fluid dynamic question, a acoustics setting or electromagnetic device, simulation can handle it.

ANSYS is a finite element analysis (FEA) code widely used in the computer-aided engineering (CAE) field. ANSYS software allows engineers to construct computer models of structures, machine components or systems; apply operating loads and other design criteria; and study physical responses, such as stress levels, temperature distributions, pressure, etc. It permits an evaluation of a design without having to build and destroy multiple prototypes in testing. The ANSYS program has a variety of design analysis applications, ranging from such everyday items as dishwashers, cookware, automobiles, running shoes and beverage cans to such highly sophisticated systems as aircraft, nuclear reactor containment buildings, bridges, farm machinery, X-ray equipment and orbiting satellites.

In this research, finite element analysis method will be applied. The retaining wall will be modeled using the ANSYS software and finite element method. Finite element method is a numerical analysis technique which is used to obtain an approximate solution for engineering problems which are difficult to solve analytically. The calculation for potential changes in design such as temperature, fluid velocities and displacement are usually very complicated. This method is the numerical analysis which is able to solve these problems.

The basic concept of finite element method is solving a continuum by a discrete model. It is done by dividing the region of interest into numerous connected sub regions or elements. Each element will be in simpler geometry and easier to be analyzed. A known physical law is then applied to each element and global equation is formed which can be used to solve the variables such as temperature and displacement.

Finite Element Analysis was applied to nonlinear problems and large deformation in late 1960s and early 1970s (Tirupathi and Ashok, 2002). In nonlinear stress analysis is becoming increasingly important with designers employing a wider variety of materials in a multitude of different applications. Nonlinear enable a design engineer to make sound design decisions. By using the nonlinear analysis in this research, it will help to predict the real behavior of the structure as it will give accurate result and we no need any more to depend on the laboratory experimental that is time and cost consuming.

2.1.1 History of Finite Element

The finite element method was introduced due to the needs for solving complex elasticity and structural analysis problems in civil and aeronautical engineering. Its development can be traced back to the work done by Alexander Hrennikoff (1941) and Richard Courant (1942). The approaches used by both pioneers are different but they share one essential characteristic whereby they disruptive a continuous domain into a set of discrete sub domains.

Alexander Hrennikoff discretized the domain by using a lattice analogy method. Meanwhile, Richard Courant's approach was to divide the domain into finite triangular sub regions in his work to study the St Venant torsion problem. Ritz method of numerical analysis and minimization of variation calculus was used by Courant to obtain the approximate solution for vibration systems. Courant's contribution was evolutionary and applied by Argyris and Kesley and Turner in their paper for the formal introduction to finite element. In the middle to late 1950s, the development of finite element began in earnest for airframe and structural analysis. The method was then provided with rigorous mathematical foundation in 1973 with the publication of An Analysis of the Finite Element Method. Since then, finite element method was widely used in engineering field.

With the rapid development of CAE technology, the finite element modeling, analysis and engineering drawings can be produced. Finite element analysis has become a vital tool in structural design and solving engineering problem

2.1.2 Nonlinear

The basic nonlinear solution approach involves a series of incremental solutions. The loads are applied in increments, load increments in a nonlinear static solution, and time increments in a nonlinear transient dynamic solution. During each increment a solution is predicted using the current state (stiffness and load increment). Depending on the type of nonlinearity, a force imbalance or residual is created during iteration where nonlinear behavior occurs. Solution iterations are required to balance equilibrium for unbalanced forces.

The iterations continue during an increment until the convergence criteria are satisfied. Once convergence is satisfied, a solution is obtained for the increment and solution progresses to the next increment using this predictor-corrector method. The detailed numerical simulation of material nonlinearity in three dimensions is still beyond the capabilities of the most powerful computers (MSC/NASTRAN for Windows, 2005).

In linear finite element analysis, all materials are assumed to have linear elastic behavior. The deformations occurred are small and assumed to have no effect on the overall behavior of the structure. Even though this kind of analysis is limited to very few real world engineering situations, but with few restrictions and assumptions, linear analysis will be sufficient for the majority of engineering applications. However, nonlinear finite element analysis is still required in certain conditions such as gross changes in structural geometry, structural cracks and permanent deformations, buckling and contact between component parts. Nonlinear finite element analysis can be divided into three types: geometry nonlinearity, boundary nonlinearity and materially nonlinearity.

2.1.3 Geometry Nonlinearity

Geometric nonlinearity occurs when the structure deforms to such an extent from the original geometry or position and the direction of load applied significantly affect the structural behavior. Common examples of geometric nonlinearity are plate structure which develops membrane behavior, or the geometric split or truss or shell structure.

2.1.4 Boundary nonlinearity

Boundary nonlinearity occurs when applied loads change a structure's boundary conditions. The changes in the boundary conditions may result from the deformation processes such as lift-off or smooth or frictional contact.

2.1.5 Materially nonlinearity

Materially nonlinearity effect occurs from the nonlinear constitutive model which has disproportionate are the plastic yielding of metal, the ductile facture of granular composites such as concrete or time dependent behavior such as creep. Material behavior can be varies depends on the current deformation state and past history of deformation. Material nonlinearities occur most in the civil engineering that deals with materials such as concrete, soils and low-strength steel.

2.1.6 Linear vs. Nonlinear

Commonly, a linear structure can sustain any load whatsoever and undergo any displacement magnitude. In the linear structure, the respond to different load system can obtain by superposition rules. Classification of linear in the cause-and effects sense. For example, if the applied forces are doubled, the displacements and internal stresses also double. Problems outside this domain are classified as nonlinear.

2.2 Truss

Truss engineering involves the design and creation of trusses that are made from one of several potential materials. Trusses are crafted from straight and slender pieces of material connected at various joints to form a triangular shape. They are primarily used to build a larger structure, with the shape and design of the truss adding strength and weather resistance to the overall structure. Trusses can be designed as planar trusses, which are a two-dimensional shape that used most commonly in roofs and floors and space trusses which are three-dimensional. Multiple trusses can also be combined.

Some truss engineering firms design and build pre-made trusses to sell to construction and architecture companies. These designs are generally smaller in scale and used for floors and roofs in houses and other relatively small scale buildings. Other firms may design their trusses and build onto the building itself instead of having them pre-made, thereby customizing the look and size to fit a specific structure.

Bridges, skyscrapers, and towers also implement the work of truss engineering professionals. More complex structures such as these require detailed planning and often a more intricate combination of trusses, comprised of multiple units being used at once. Bridges and towers generally combine planar and space trusses that are made from metal, although some bridges are made of wood.

More detailed aspects of truss engineering involves the design of multiple trusses that are interwoven in a particular shape or for a specific purpose. Generally, the more complex the design, the stronger the structure it will be. This is particularly important for the tops of skyscrapers and other tall buildings because they must withstand much stronger wind resistance than structures that are lower to the ground.

In building construction, a structural frame usually fabricated from pieces of metal or timber to form a series of triangles lying in a single plane. The linear members are subject only to compression or tension. The horizontal pieces forming the top and bottom of the truss are called the chords, and the sloping and vertical pieces connecting the chords are collectively called the web. Unlike a vault, the truss exerts no thrust but only downward pressure; supporting walls require no buttressing or extra thickening. Trusses have been used extensively in roofing and bridges.

2.2.1 Type of truss

In architecture and structural engineering, a truss (or braced framework) is formed when members are put and connected together at joints (or nodes). It is comprised by one or more triangular units together with straight slender members. The figure 2.1 below shows some common type of truss:

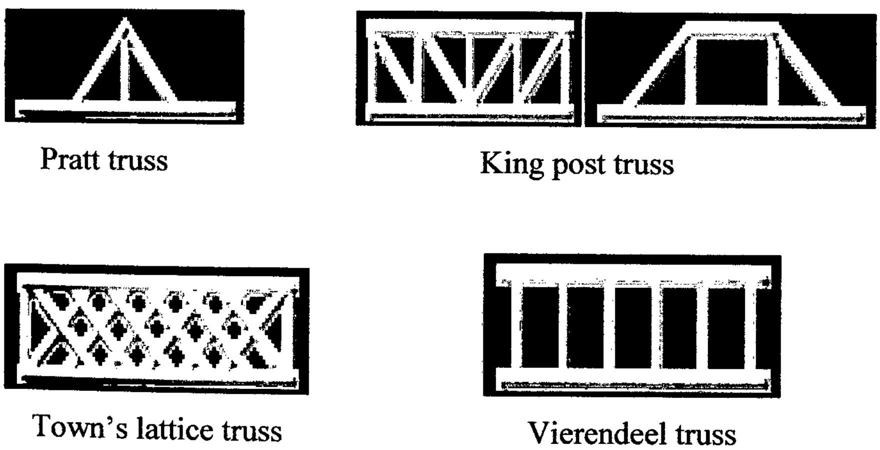

Figure 2.1: Types of truss

There are many types of truss such as Pratt truss, Bow string truss, King post truss, Lenticular truss, Town's lattice truss, Vierendeel truss and many others. However, truss analysis can be carried out by assuming the truss as a planar truss or space truss. Planar truss or Plane truss is a truss where all the members and nodes are lying within a two-dimensional plane. Space truss is a truss that having members and nodes extending into three dimension.

In most cases in the analysis of truss, a truss is usually modeled and analyzed as a two-dimensional plane frame. At the same time, if there are significant forces out-of-plane, the truss should be modeled as a three-dimensional space for further analysis. In structural analysis, there are several methods that can be used to analysis truss which are method of joints, method of sections, graphic statics, finite element method and etc.

The method of joints uses the equilibrium of joints and becomes the basis for all trusses analysis directed towards finding the unknown forces in the truss structure. A truss is considered to be composed of a series of members and joints. Member forces are found considering all the joints or points in a state of equilibrium. For plane trusses, the two independent equations of statics exist for a concurrent force system ($\Sigma Fx=0$ and $\Sigma Fy=0$). However, the joint will move if there was a net force acting to a joint.

The method of sections is a method which cut the truss structure into sections, then replace the removed section with unknown member forces acting in the direction of the cut member. The forces in the members are then computed by summing the unknown forces by using equilibrium equation, $\Sigma Fx=0$, $\Sigma Fy=0$ and $\Sigma M=0$. Since there are only three equilibrium equations, the truss section cut should be located where there are only three unknown member forces.