DESIGN ANALYSIS OF UNBRACED FRAME USING ANSYS

FARHANA BINTI ABDUL HALIM

Thesis submitted in partial fulfillment of the

requirement for the awards of the degree of

Bachelor of Civil Engineering

Faculty Of Civil Engineering And Earth Resources

Universiti Malaysia Pahang

NOVEMBER, 2010

ABSTRACT

Recently, the application of unbraced steel frame in building construction arises rapidly nowadays, various researches has been carried out to study the behavior of this type of frame. The tested frame structure was designed using available recommendation in the existing literature. Finite Element Analysis is developed to stimulate unbraced frame's deflection behavior. The result is compared with the range of value of several goodness-of-fit measures which are provided from random output parameter in each response surface set. Polynomial mathematical model generated using various combination parameters using multiple regression analysis were found to be statistically significant. Parametric study using response surface methodology through finite element analysis may form efficient approximation to immediate deflection. It shows a comparable result and thus, this program is expected applicable.

ABSTRAK

Dewasa ini, aplikasi kerangka tidak dirembat sebagai struktur utama bangunan semakin pesat membangun dari semasa ke semasa, pelbagai kajian telah dilakukan bagi mengkaji sifat-sifat kerangka jenis ini. Ujian terhadap struktur kerangka ini telah diambil dari struktur yang direka bentuk oleh kajian literasi sebelum ini. Kaedah Elemen Unsur Terhingga telah dibangunkan untuk meransang sifat lenturan kerangka tidak dirembat. Keputusannya akan dibandingkan dengan anggaran nilai dari beberapa ukuran 'goodness-of-fit' yang mana terhasil dari hasil keluaran parameter yang rawak di setiap set tindakbalas permukaan. Model matematik polinomial telah dihasilkan oleh pelbagai gabungan parameter menggunakan analisis kemerosotan pelbagai yang dijumpai dan menjadi statistik penting. Kajian parametrik menggunakan kaedah tindakbalas permukaan melalui elemen unsure terhingga boleh menghasilkan pesongan yang hampir benar. Ini menunjukkan perbandingan antara hasil keputusan dan program ini dianggap boleh diterima guna.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 General

Recently, the need for structural rehabilitation of civil infrastructures all over the world is well known and a great amount of research is going on this field. Every time new technology will develop which are changes in the contributing factors to structure such as increase in load requirement, corrosion deterioration to exposure to aggressive environments, changes in functionality, potential damaged caused by mechanical and environmental effects, increase in material strength and durability, etc.

Many different methods are suitable for repair and strengthening, such as additional reinforcement cover by concrete, external steel plate bonding, using of steel material, etc. Nowadays, steel frame system with beams and columns is become the conventional building structure in construction world. Structural design and structural

analysis are both of the criteria needed to create a structure that safely accomplish its function in order to produce structures in a stability condition. In civil engineering field, steel is widely used in building construction. Its popularity may be due to the various sizes and the shape of steel sections to be used for various type of structures such as small and the simple buildings as well as complicated infrastructures construction. Generally, steel frame not is only design to sustain vertical loads but also able to resist lateral loads.

In this research, Finite Element Method (FEM) models were used to stimulate the behavior of the steel frame structure's ability using ANSYS 12.1 program. The ANSYS was founded in 1970, develops and globally markets engineering simulation software and technologies widely used by engineers and designers across a broad spectrum of industries like civil and mechanical engineering. This program is capable of predicting deflection and stress in concrete concepts and also includes model's constitutive laws for concrete material, based on smeared crack concepts and for high strength composite materials.

1.2 Problem Statement

Columns and beams which are in parts of frame structure are also known as main components in building which have their own capability to support the load from roof, floors and transfer it to lower level and foundation. Because of that, frame structures must be designed well with appropriate calculation to make sure it can support the loads so that it is safe for long period of time. In order to make the calculation more easily but still in good design method of the structure, therefore ANSYS Parametric Design Language is use to generate a new equation for calculating the design of steel frame. The new equation is derived from parameters which are depends on loads, length of span, Poisonn's ratio, pressure on structure and so on. Therefore, this is the necessary of this research.

1.3 Objective

The objectives of this research are following:

- To present software analysis with analytical results.
- Generate a new equation for frame structure by proving the parameters in ANSYS Parametric Design Language (APDL) using ANSYS program.
- To study the response and behavior of unbraced frame structure through a series of analysis under different load and dimension case.

1.4 Scope of Work

These researches are mainly focused on the design of steel frame structures by using British Standard and use the data to generate the new equation by proving the parameters in ANSYS. In order to achieve the objectives of the researches, there are few researches scope is necessary to be followed. Study the types of steel design frames and the characteristics of the structure.

Explore the ANSYS program by learning how to use the programming by using tutorial from internet. Practicing of tutorials can helps to solve problem when running the real models. Model of columns are designed using ANSYS parametric design language (APDL). Finite Element Analysis is using to calculate the parameters which the parameters must success to get new equation as mention in objectives.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

In recent years, numbers of studies have been made in designing the unbraced multi-storey steel frames and the development in this type is still continued until now. Many researchers try to present the easier and simple way for example comes out with software such as ANSYS, Visual Basic and so on for determination of the section sizes which meet the principals and limitations as stated in BS 5950-1:2000.

Apart from that, several methods were introduced to fulfill the criteria which are Wind-Moment Method and Merchant-Rankine Method. However, the effectiveness of these methods is still in question because the second order effect due to sway does not taking into account though sway deflection give the major effect in

the construction of frames. It is irrelevant to attempt final design that exceeds the sway limitation because sway does not affect the frame structure. Therefore, a simplified method which meets the sway deflection criterion is introduced and the method is called as Direct Design Method.

This method does not stand alone. A supporting computer program should be established to help the researchers reducing the difficulty when designing the frame structure. The new software among students in Malaysia that can be used together to compared with manual calculation is ANSYS 12.1. The establishment of this computer program actually can help us to modify the structure's design and more easy if it does not meet the specification as stated in BS5950-1:2000.

2.2 Frame

Frame is a structure made up from columns and beams that connected to each others at a joints. It can be classified into two types which are braced frame and unbraced frame. The behavior of these two frames is different to each other when subjected to the lateral loadings.

2.2.1 Braced frame

Braced frame is a frame that has been supported by a bracing system to prevent it from sway when subjected by lateral loading to meet the requirement. The installation of bracing system had provided more stability to the system because it can resist the lateral loadings from winds and earthquake or in other words, lateral loadings did not taken by the column and beam (Schodek, 2004). For this type of

frame, it only took vertical loadings from dead load and live load. Figure 2.1 shows the bracing system used in steel frame construction.

2.2.2 Steel frame

The complexity of analyzing and designing of steel frames have decreased the use of steel material in building construction. However, many approaches have been made to study the feasibility of steel frames structure since 1856 by Besemer's process. Nowadays, many tall building is used the steel frame structure.

Frame structure can be classified into two types, braced and unbraced frames. Braced frame which consist of bracing system that will provide lateral stability to the structure, so, it will be resistance to the lateral effect. On the other hand, unbraced frame does not provided with the bracing system and will easily sway if the lateral loads act on it.

The lateral loading normally cause by the wind forces or earthquake. When these lateral forces acting against the structure, it will induce overturning moment and should be balanced by the internal moment developed by the members. Therefore, the frames should be design properly to ensure that it will be able to carry both lateral and vertical loads.

2.4 Loads On Structure

There are many types of loading that can affect the behavior of the structures. When designing a building, those loadings should be taken into consideration because if it is miscalculate, absolute failure will occur in the structure. Figure below illustrates and describing the types of loads that must be considered (Atiliana, 2009).

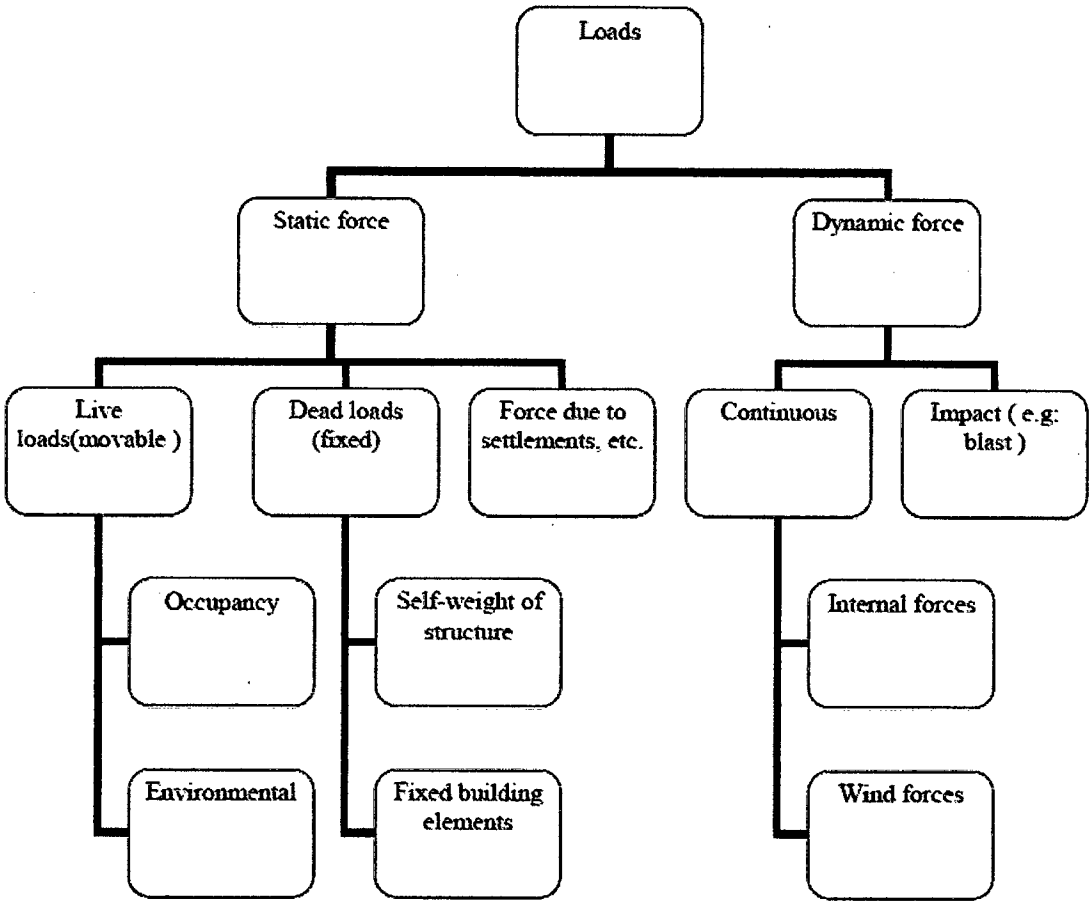

Figure 2.2 Typical types of loading condition

2.4.1 Static force

The static forces consist of live load, dead load and forces due to settlement or thermal effect. These types of load are typically being considered when designing every structure.

Live loads are forces that act vertically downward onto the structure but it is not fixed in character. The value can change anytime and movable. Occupancy and environmental can be described as live loads. Occupancy loads include furniture, stored material, human or any other similar material.

On the other hand, dead loads are non-movable loads and fixed in behavior. There are several items that can be marks as dead loads and those are self-weight of the structure, slab, bricks, mechanical equipment or any other building.

2.4.2 Wind loads

This type of loading is one of the most important characteristic especially for multi-storey buildings because it can cause sway to the structure. As the wind move, it will deflect or stop when something blocking its path. This phenomenon will cause the kinematic energy transform into potential energy. This potential energy is called force that induced horizontally onto the structure. Therefore, high-rise building will be design properly to resist this force.

2.5 Finite Element Analysis

Finite element analysis (FEA) is consists of 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. In other words, FEA is a numerical method to find out an approximate solution for variables in a problem which is difficult to obtain analytically. The calculation of potential design changes such as temperature, buckling and deflection are usually complicated. A numerical method that is able to solve these engineering problems id the finite element analysis. In case of structural models failure, FEA may used to help determine the design modifications to meet the new condition.

The concept of the finite element analysis is solving a continuum by a discrete model. It is done by dividing the problem into small several elements. Each element is in simple geometry and this is easier to be analyzed than the actual problem or the real structure. Each element is then applied with known physical laws. The equation which is formed by each element or parameters then will combined to form a global equation. The new equation can be used to solve the field variables such as displacement, buckling, temperature and so on.

The aeronautics, automotive, defense, and nuclear industries had started using the finite element application since early 70's. However, this is limited to expensive mainframe computer. Zienkiewicz and Cheung is the important person in developing the finite element technology at that time. But later, Hinton and Crisfield carried out the finite element into modeling and solution of nonlinear problems (Reddy, 1993).

With the development of the CAE technology, engineering drawing can be produced. Besides, the analysis can be carried out and also the Finite element modeling can be done. The finite element becomes more and more important today which can simplify and solves various types of engineering problems.

2.5.2 How Does Finite Element Analysis Work

FEA uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements (Peter Widas,1997).

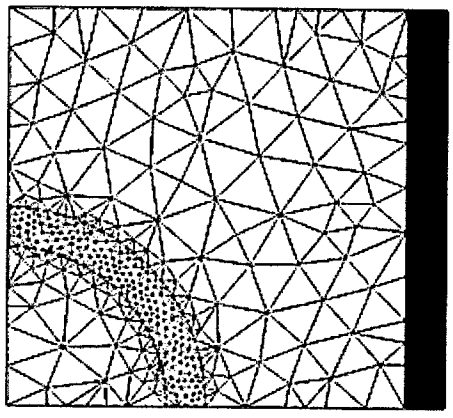

Figure 2.4 Mesh diagram on 2D model

The powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications. The FES has substantially decreased the time to take products from concept to the production line. It is primarily through improved initial increased accuracy, enhanced design and better insight into critical design parameters, fewer hardware prototypes, a faster and less expensive design cycle, increased productivity (Peter Widas,1997).

2.5.3 Meshing

The important requirement of the FEM is the need to split the solution domain (model geometry) into simply shaped subdomains called 'finite elements'. This is a discretization process commonly called meshing and element are called finite because of their finite, rather than infinitesimally small size having infinite numbers of degree of freedom. Thus the continuous model with an infinite number of degrees of freedom (DOF) is approximately by a discretized FE model with a finite DOF. This allows the reasonably simple polynomial functions to be used to approximate the field variables in each element. Meshing the model geometry also discretized the original continuous

APDL are can be known as tools to help designers perform parametric analyses in which simulation software automatically solves for entire ranges of specified variables and generates displays that enable users to readily spot trends and identify an optimal design. Parametric analysis can guide the product development process to a design configuration that can save the time compared if by individual analyses were manual performed. The period of time to set up is required in APDL which a model can increase significantly with the complexity of the geometry.

ANSYS Workbench platform is one of the most efficient ways of deal with geometric parameters which enables parameters of the CAD model to be driven directly from simulation.

2.6.1 Benefit of parametric design analysis

Parametric analysis is an excellent to get accurate information about the influence of all parameters on the design objectives, such as system performance with respect to stress, loading, deflection of structure and other variables. With this information, the design team can make informed decisions throughout product development. In addition, the design teams also can response quickly to any modification due to external constraints for example, manufacturing (Thieffry, 2008).

CHAPTER 3

METHODOLOGY

3.1 Introduction

Nowadays, research undertaken the finite element analysis was proven to be a successful tool in predicting the behavior of frame structures. From the previous research, finite element analysis provides only approximate solution and the results are not fully can be used in real design with reasonable confidence. This is because there are various ways to perform the finite element process such as meshing of structure which varied analysis results may be obtained. Therefore, the comparison between manual calculations and finite element analysis is needed to determine their efficiency and reability.