ANALYSIS OF UNBRACED FRAME USING ANSYS

LEE YING YING

Thesis 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

JUNE 2012

ABSTRACT

Construction of structures using steel as the construction material has become one of the major alternatives to the conventional reinforced concrete. Frame is one of the important components for a structure and need to be analyzed 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. Among the various numerical methods, finite element analysis is very popular and widely used for solving engineering problems. Finite element analyses already become an integral part of computer aided engineering. ANSYS is a general purpose finite element modeling for numerically solving wide variety of structural analysis problems which include static structural analysis for both linear and non linear and heat transfer. 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 frame finite element analysis software using ANSYS. The objective of this study is analysis the strength behavior of unbraced frame under various loading which include the gravity and wind load. A step-by-step Finite Element formulation of unbraced frame 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

Besi merupakan bahan binaan yang amat terkenal dalam bidang pembinaan struktur pada masa sekarang. Kekanda 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. Antara cara kajian, analisis kaedah unsur terhingga dikatakan merupakan suatu cara yang sesuai untuk menyelesaikan masalah kejuruteraan awam. Kaedah unsur telah dijadikan satu bahagian daripada analisis computer. ANSYS merupakan satu program yang mengunakan kaedah unsur yang digunakan untuk menyelesaikan masalah struktur. 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 pembangunan perisian untuk menjalankan analisis element unsur terhingga terhadap kekuda dengan menggunakan ANSYS. Objektif kajian ini adalah untuk mengenal pastikan kekuatan kekanda di bawah barat gravity dan angin 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 kekanda dibandingkan dengan keputusan dari STAAD.Pro untuk membuktikan kebolehannya.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 General

Construction of structures using steel as the construction material has become one of the major alternatives to the conventional reinforced concrete. Since the introduction of Bessemer's process in 1856, many approaches and researches have been made to study the feasibility of steel frames. Effect of various loading condition towards the frame has become the concern issues for most of the researcher. Buckling and deflection is the common effect that affect by the loading.

Multi storey frames divided into two distinct categories for the purpose of design: braced and unbraced frames. Braced frame is a frame consisting 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 lateral load act on it. There are two methods of design and construction of steel frame where are simple construction (brace frame) and rigid construction (unbraced frame). In simple construction, the beams are connected to columns by pin or simple joints, which are designed to carry moment. The beams are therefore analyzed and designed as simply supported beam. In this type of construction, stability of the structure is given by the diagonal bracing members. Columns are subjected to axial loads and nominal bending moments due to the beam reactions that act eccentric to the neutral axis. In rigid construction, the beams are connected to columns by rigid joints which are designed to carry bending moment. The beams and columns are analyzed as continuous frame.

Among the various numerical methods, finite element analysis is very popular and widely used for solving engineering problems. Finite element analyses already become an integral part of computer aided engineering. ANSYS is a general purpose finite element modeling for numerically solving wide variety of structural analysis problems which include static structural analysis for both linear and non linear and heat transfer. In general, a finite element solution may be broken into three stages which are preprocessing, solution, and post processing. In preprocessing stage, model will build to define key points, element type and material, and mesh lines as required. While for solution stage, the loads and constraints will specify and finally solve the resulting set of equations. In post processing stage, further processing and viewing of the results will see such as the lists of nodal displacements, element forces and moments, deflection plots, and stress contour diagrams.

In this study, a 4 storey building with unbraced frame is used to analyze the deflection and stress of frame. Finite element method, (FEM) is using in this study in order to get a more accurate result for the behavior of frame. The school building is modeled out using ANSYS in order to analyze the behavior of frame under various conditions. Various loading condition include vertical load (e.g. uniformed distributed load) and horizontal load (wind load) is apply to the frame. Contour diagram of deflection and stress distribution result of the analysis is clearly shows and stated in ANSYS. Besides, probabilistic analysis is doing in multiple simulations.

1.2 Problem Statement

Loads applied to structural members during construction quite often exceed the design service loads that will act on the completed building. In December 1985, a catastrophic accident occurred in Los Angeles during construction of a 21-story steel-frame building. Eighty tons of structural steel sections were stockpiled on one bay on the fifth floor, loading the bay to twice its designed capacity. Three beams failed suddenly at their temporary connections, precipitating a progressive collapse of 10 bays all the way to the ground floor.

Variable load act to the frame become one of the factor that cause frame fail. Vertical load and horizontal load that acting to the frame structure are affect the frame bahaviour. Frame that cannot resist the loading act on it will reach the failure mode. Buckling and collapse of unbraced steel frame are the most common failure case.

Finite element method is very popular and widely used method for solving engineering problems. Compare to conventional method, FEM give accurate solutions in analyses almost any structure having any shape and made of any material. Simple linear static problem as well as highly complex nonlinear transient dynamic problems is effectively solved using the finite element method. Also, in code of practice (i.e. BS5950) does not provide provision about the behavior of unbraced frame.

ANSYS is use to define the strength behavior of the columns by modeling the structure. It is a program which can solve variety of structure engineering problems by showing the deflection and stress of the frame. If compared to Esteem Plus program, ANSYS have more advantage in determine the strength behavior of the structure. Generally, Esteem Plus program does not provide any contour diagram about the deflection and stress of the frame. Besides, effect to strength behavior of column also cannot derive in Esteem Plus program.

1.3 Objectives

The objectives of the study are listed below:

1. To conduct a study on steel frame using finite element method. Stiffness matrix method of finite element analyses can give more accurate solution on frame.
2. To model the frame structure using ANSYS program. ANSYS program will used to model the steel frame. Preprocessing, solution, and post processing are three stage that need to discover in this study. Parameters can find out and get from the ANSYS probabilistic design method.
3. To study the factor that will affect the behavior of unbraced frame through a series of analysis. Effect of wind load, selftweight beam, weight of slab will consider as the load acting to the frame.

1.4 Scopes of study

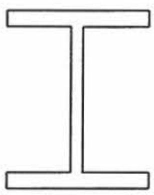
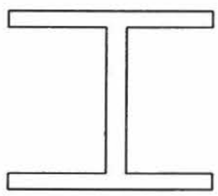
In order to achieve the objectives of the study, learning how to use the ANSYS program through internet is start. A few tutorials in website University of Alberta is doing to practice. Practicing of tutorials can help to solve problem when running the real models. Model of frame are designed using ANSYS parametric design language(APDL). Finite element analysis is using to calculate the parameters which the parameters must success to get result.

This study will focus on the effect of the various load that acting on the unbraced frame using ANSYS. A series of analysis will run to get the deflection and stress distribution. A report of APDL will obtained to know the parameter. Table below show the properties of the unbraced frame.

Table 1.1: Geometric properties of unbraced frame

Properties	Description
Type of frame	Unbraced frame
Type of steel	Hot rolled steel
Elastic modulus(N/m^2)	200000
Poisson ratio	0.3

Table 1.2: Material properties of beam and column section

<table>
<tr><th>Details</th><th colspan="2">Beam</th><th colspan="2">Column</th></tr>
<tr><td>Section</td><td colspan="2">305 x 105 x UB40</td><td colspan="2">254 x 254 x UC73</td></tr>
<tr><td>Crossectional area</td><td colspan="2">51.3cm^2</td><td colspan="2">92.9cm^2</td></tr>
<tr><td>Moment of inertia</td><td colspan="2">8500cm^4</td><td colspan="2">11360cm^4</td></tr>
<tr><td>Mass per metre</td><td colspan="2">40.3kg/m</td><td colspan="2">73kg/m</td></tr>
<tr><td rowspan="5">I section properties</td><td colspan="2"></td><td colspan="2"></td></tr>
<tr><td>Width of section, B</td><td>165mm</td><td>Width of section, B</td><td>254mm</td></tr>
<tr><td>Depth of section, D</td><td>303.4mm</td><td>Depth of section, D</td><td>254mm</td></tr>
<tr><td>Thickness flange</td><td>10.2mm</td><td>Thickness flange</td><td>14.2mm</td></tr>
<tr><td>Thickness web</td><td>6.0mm</td><td>Thickness web</td><td>8.6mm</td></tr>
</table>

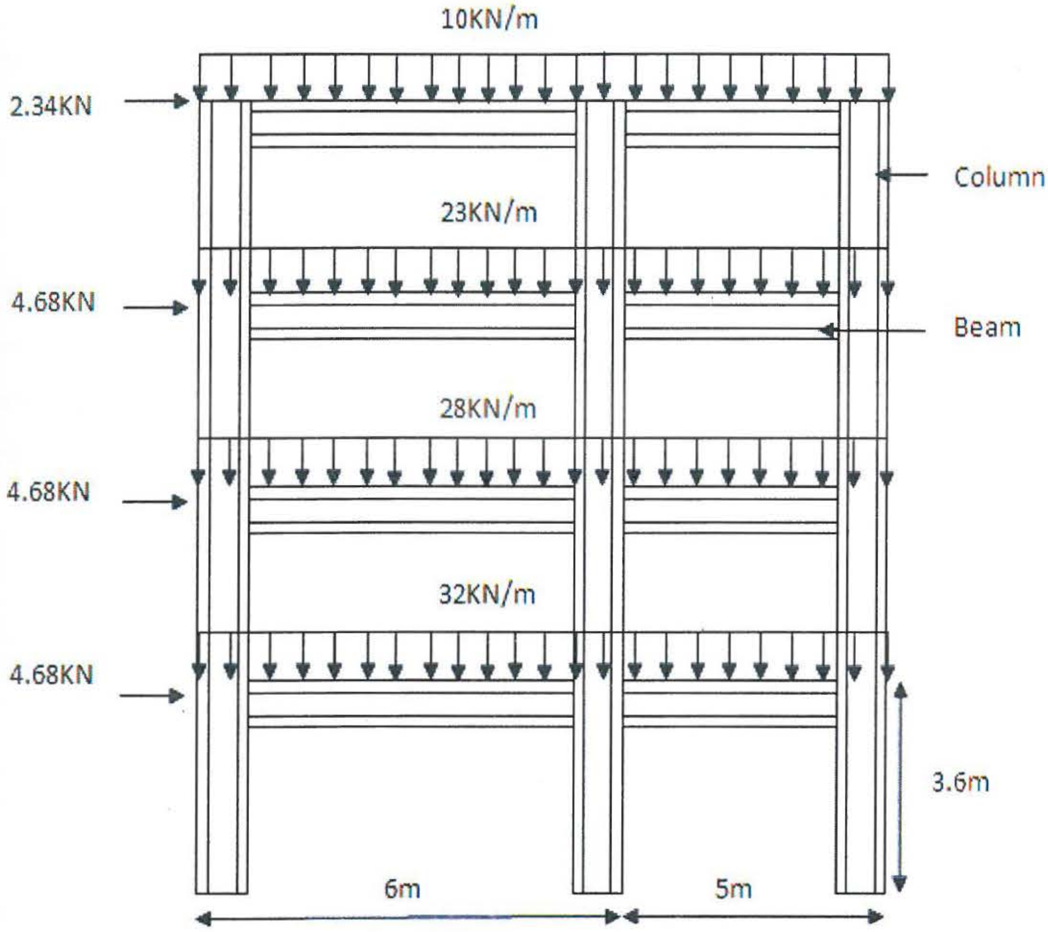


Figure 1.1: Unbraced frame with loading.

1.5 Significance of study

It is necessary to have knowledge about the behavior of unbraced frame under various conditions. Analysis is done to know the critical of the frame behavior under the effect of load, buckling, and temperature. The main purpose of the study is to stimulate the behavior of frame by using Finite Element Method. Besides, factor affect the stability of unbraced frame is determined using APDL analysis.

This study can be as a reference for further studies and further improvement for related search in the future. By identify the strength behavior of frame using finite element analysis, this would make a good investment. Strength behavior is important in order to avoid any structural failure of the building.

This study can proved that finite element formulation gives more accurate results in the global behavior of the frame system. As the actual model is replaced by a set of finite elements, this method gives an approximate solution rather than an exact solution. Besides, this study can determine that ANSYS is suitable program to overcome the structure strength behavior problems.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The structural use of steel in the construction industry is continually growing rapidly across world. The use of steel as a construction material has its advantages, such as strength, lightness, ductility, etc. It also considers challenges with regard to slenderness, stability, fire resistance, geometric imperfections and other structural requirements. In recent years, numbers of studies have been made in analyzed the unbraced steel frames and the development in this type of structure is still continued until now. Many researchers try to present the analyses which meet the principal and limitations in their research.

2.2 Frame analysis

Several methods were introduced to fulfill the criteria. Those methods are Wind moment method and merchant Rankine method. For unbraced frames, early consideration needs to be given to limiting sway in service conditions. D.Anderson (2006) stated that the analysis of unbraced multi storey steel frames is a particular example of such endeavor, which has resulted in the wind moment method and the

Merchant Rankine formula continuing to be used in design. For wind moment method, the method assumes under gravity loads the beam to column joints act as pinned connections. Under horizontal wind loads these joints are rigid. Points of contra flexure are assumed to occur at the mid height of columns and the mid length of beams. In contrast, the Merchant–Rankine formula allows as full a distribution of plastic hinges as is possible when account is taken of second-order effects. The load level at failure, λ_f, is assumed to be given by:

$$1/\lambda_f = 1/\lambda_{pl} + 1/\lambda_{cr}$$

Where λ_{pl} the load is level for rigid-plastic collapse and λ_{cr} is that for elastic critical buckling. For rigid-jointed frames, a modified relationship, making some allowance for strain-hardening and stray composite action:

$$1/\lambda_f = 0.9/\lambda_{pl} + 1/\lambda_{cr}.$$

For $\lambda_{cr}/\lambda_{pl} \geq 10$, second-order effects are ignored and the failure load is taken as that for rigid-plastic collapse. Limiting values of $\lambda_{cr}/\lambda_{pl}$ have been used in design codes.

As part of a study to investigate extending the scope of the Wind Moment Method (WMM) for the design of unbraced frames to cover composite construction, another comprehensive body of numerical data on the performance of a series of representative frames has been generated by J.S Hensman, D.A Nethercot(2001). The response of a large number of frames, designed using the proposed WMM approach, was compared with the set of design requirements. The actual non-linear behaviour of the beam-to-column connections and the semi-rigid behavior of the column bases were allowed for in all the analyses. This has required the development of a suitable numerical modeling technique such that all important effects within the frames could be adequately represented.

Besides wind moment method and the Merchant Rankine formula, a modified Muto;s method is developed by Y.L. Wong, T.Yu, and S.L.Chan in July 2007. This method used to analyze unbraced bare steel frames with semi rigid connections under working load conditions, and to modify the equivalent beam stiffness of a composite beam. Unbraced frame can model with semi rigid connections. This method can be used to qualify the effects of semi rigid connections on the inter storey drifts of composite frames under lateral loads.

In this study, an unbraced frame is used to analyze. Wind moment method of frame analysis is considered. Wind load is calculated based on the standard of Code of practice on Wind Loading for Building Structure. The structure is rigid construction where the beams are connected to columns by rigid joints which are designed to carry bending moment. The beams and columns are analyzed as continuous frame.

2.3 Frame

A multi-storey building must resist the combined effects of horizontal and vertical loads; it is composed of foundations, frameworks and floor slabs. Frame is a structure made up from columns and beams that connected to each other at the joints. It stabilizes the building by resisting horizontal actions (wind and seismic loads). Floor slabs are supported by beams so that their vertical loads are transmitted to the columns. They are made of reinforced concrete or composite slabs using profiled steel sheets. Columns are commonly made of H or hollow hot-rolled steel sections. The use of hollow sections filled with concrete can improve their fire resistance. Beams are commonly made of I and H profiles. Nevertheless, the use of welded built-up sections can offer more rational solutions in some cases.

Frame can be classified into two types which are braced frame and unbraced frame. The behavior of these two frames is differs to each other when subjected to the lateral loadings.

Braced frame is a frame that has been supported by a bracing system to prevent it from sway when subjected by lateral loadings 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. For this type of frame, it only took vertical loadings from dead load and live load.

For unbraced frame, it does not have any bracing system that can prevent it from sway when lateral loadings act on it because it takes all the axial force, shear force and bending moment. Unlike braced frame, unbraced frame must design properly so that it can resist the loadings that can cause it sway. Several criteria need concern when designing unbraced frames such as sway limitation, relative stiffness of the column, type of connection, etc.

2.4 Buckling mode

For long compression members, it will tend to change the shape and become unstable when force acting onto the members. When this happen, it means that the loading already exceed the limit and the members slowly lost the capability to carry the loading. This pheneomenon can be called as buckling. Buckling can also be defined as a sudden failure of a member when subjected to a compressive stress and the actual stress is less than the ultimate compressive stress that the structure is capable to withstand at the failure point.

Regard to this, a research is done by L.Xu and X.H.Wang in evaluating the stability of multi storey unbraced steel frames subjected to variable loading. In the case of variable loading, the conventional assumption of proportional loading is abandoned, and different load patterns may cause the frame to buckle at different levels of critical loads. In my study, variable load such as wind load and distributed load will consider to analyze the frames. This done by using ANSYS to check and evaluate the buckling of frames after the load is applying on it. Stability of the frame scan clearly sees in this result shown in ANSYS.

Ceng Shu Tong and Lei Zhang (2008) have presented steel frames where each storey is treated as a whole at the sway buckling, its resistance is evaluated using the relationships illustrated by the column curves to examine the proposed method. The interaction between the sway buckling of the whole storey and the non-sway buckling of the weakest column may reduce the failure load of steel frames. In this study, a reduction factor is presented to consider this mode interaction, and comparisons with FE results show that this method can reasonably assess the adverse effects of this mode interaction. Comparisons between FE results show that the presented method has good performances in predicting the failure load of considered frames. In FE analysis, the geometric and material nonlinearities as well as two types of geometric imperfections and longitudinal residual stresses are considered. Predictions from the notional load approach are also compared with FE results for some cases, which show that these predictions may underestimate or overestimate the failure load of certain frames.

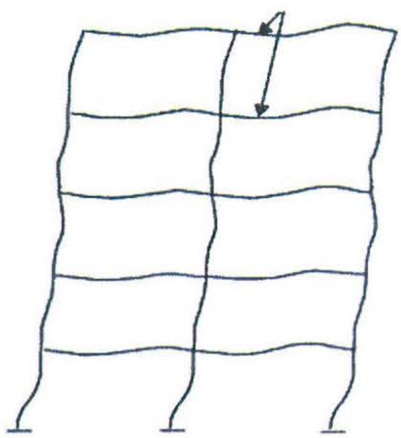


Figure 2.1: Critical buckling mode for an unbraced frame