



NONLINEAR FINITE ELEMENT ANALYSIS OF STEEL FRAME

NADIA ASHIKIN BINTI ZAINAL BAHRI

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

JUNE 2012

## 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 steel frame structure using Ansys 12.0 (APDL) and determines the maximum deflection 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. There are six random inputs will be used in this study which is Young Modulus (EX), Poison Ratio, Density, Area, Uniform Distributed Load (UDL) and Fy Loads. The random output will be the graph of maximum deflection. This study is going to be basic guidelines on how to implement the civil engineering element using Ansys software. This paper considers the practical application of nonlinear models in the analysis of steel structures. The results of some analyses performed using the steel model of the general purpose finite element code Ansys are presented and discussed. The differences observed in the response of the same steel frame as some variations are made in a material model that is always basically the same are emphasized. The consequences of small changes in modeling are discussed and it is shown that satisfactory results may be obtained from relatively simple and limited models.

## ABSTRAK

Analisis unsur terhingga (FEA) telah menjadi perkara biasa sejak beberapa tahun, kebelakangan ini, dan sekarang menjadi asas kepada berbilion nilai industri setahun. Penyelesaian berangka walaupun untuk masalah tekanan yang sangat kompleks sekarang ini dapat di selesaikan dengan mudah menggunakan FEA menjadikan kaedah ini sangat penting. Kajian ini merangkumi pemodelan rangka bingkai menggunakan perisian Ansys 12.0 (APDL) dan menentukan nilai maksimum pesongan tekanan apabila beban di kenakan. Ansys adalah perisian berasaskan FEA yang di gunakan meluas di dalam bidang kejuruteraan perkomputeran. Enam data rawak akan dimasukkan iaitu Young Modulus (EX), Poison Ratio, Ketumpatan, Luas, Beban Agihan Seragam (UDL) dan beban menegak. Kajian ini akan menjadi rujukan asas untuk menerapkan unsure kejuruteraan awam menggunakan perisian Ansys. Kertas ini berpendapat permohonan praktikal model tak linear dalam analisis struktur besi tetulang. Keputusan beberapa analisis yang dilakukan menggunakan model besi bertetulang untuk kegunaan am yang terhad Ansys kod elemen dibentangkan dan dibincangkan. Perbezaan yang di perhatikan dalam sambutan rasuk besi bertetulang yang sama kerana beberapa variasi yang di buat dalam model penting yang sentiasa pada dasarnya sama menekankan. Akibat perubahan kecil dalam pemodelan di bincangkan dan ia menunjukkan bahawa keputusan yang memuaskan boleh diperolehi daripada model yang agak mudah dan terhad.

## TABLE OF CONTENT

<b>SUPERVISOR'S DECLARATION</b>	ii
<b>STUDENT'S DECLARATION</b>	iii
<b>ACKNOWLEDGEMENTS</b>	v
<b>ABSTRACT</b>	vi
<b>ABTRAK</b>	vii
<b>TABLE OF CONTENTS</b>	viii
<b>LIST OF TABLE</b>	xii
<b>LIST OF FIGURE</b>	xiv
<b>CHAPTER 1          INTRODUCTION</b>	
1.1    Background of Study	1
1.2    Problem Statement	3
1.3    Objectives	3
1.4    Scope of Study	3
1.4.1    Finite Element Analysis (FEA) Software	4
1.4.2    Analysis Process for Stéel Frame Shelter	4
1.4.3    Selection of An Available Steel Frame wills be used in this project	4

1.4.4	Boundary Condition of Steel Frame	5
1.4.5	Simulation and Analyzing process of steel beam using ANSYS	5
1.5	Significant of Study	5

## **CHAPTER 2            LITERATURE RIVIEW**

2.1	Introduction	6
2.2	Shelter Structural Analysis Using ANSYS	7
2.3	Nonlinear FEA Analysis	8

## **CHAPTER 3            METHODOLOGY**

3.1	Introduction	11
3.1.1	Get all data of the selected structure	11
3.1.2	Modeling the structure (steel frame)	12
3.1.3	Simulation the model using the software	13
3.1.4	Analyze the data	13
3.1.5	Make conclusion based on the research been made	13
3.2	Flow Chart	13
3.3	Modeling	15
3.3.1	Define the type of element	19
3.3.2	Define geometric properties	20
3.3.3	Element material properties	21
3.3.4	Mesh size	22

3.3.5	Mesh	22
3.3.6	Saving the work	22
3.3.7	Define analysis type	23
3.3.8	Set solution control	23
3.3.9	Apply constraints	24
3.3.10	Apply loads	24
3.3.11	Solve the system	24

## **CHAPTER 4            RESULTS AND ANALYSIS**

4.1	Introduction	25
4.2	Deterministic Model Information First Model	26
4.3	Probabilistic Model Information	27
	4.3.1 Random input variables	28
4.4	Probabilistic Analysis Summary	34
	4.4.1 Solution set nonlinear steel frame analysis	34
4.5	Probabilistic Analysis Results	35
	4.5.1 Statistic of the probabilistic results	35
	4.5.1.1 Result set nonlinear steel frame analysis	35
	4.5.2 Sample history plots	37
	4.5.2.1 Result set nonlinear steel frame analysis	38
4.6	Histogram Plots	43

4.6.1	Histogram of input variables	43
4.6.2	Histogram of output variables	50
4.7	Cumulative Distribution Function Plots	51
4.7.1	Cumulative density functions of input variables	51
4.7.1.1	Result set nonlinear steel frame analysis	52
4.7.2	Cumulative density functions of output variables	54
4.7.2.1	Result set nonlinear steel frame analysis	54
4.8	Sensitivity Plots	56
4.8.1	Result set nonlinear steel frame analysis	56
4.9	Correlation Coefficient Matrix	58
4.9.1	Result set nonlinear steel frame analysis	58

## **CHAPTER 5 CONCLUSION AND RECOMMENDATION**

5.1	General	62
5.2	Conclusion	62
5.3	Recommendations	63

<b>REFERENCES</b>	<b>64</b>
-------------------	-----------

## LIST OF TABLE

<b>Table No.</b>	<b>Title</b>	<b>Page</b>
3.1	Coordinate 1	16
3.2	Coordinate 2	17
3.3	Coordinate 3	18
4.1	Details of the Finite Element Model	27
4.2	Material Properties	27
4.3	Random Input Variable Specifications	28
4.4	Specifications for Monte Carlo Simulation Analysis	34
4.5	Result Set Nonlinear Steel Frame Analysis-Statistics of the Random Input Variables 1	36
4.6	Result Set Nonlinear Steel Frame Analysis-Statistics of the Random Input Variables 2	36
4.7	Result Set Nonlinear Steel Frame Analysis-Statistics of the Random Output Variables	37
4.8	Result Set Nonlinear Steel Frame Analysis-Linear Correlation Coefficients between Input Variables 1	58
4.9	Result Set Nonlinear Steel Frame Analysis-Linear Correlation Coefficients between Input Variables 2	59



4.10	Result Set Nonlinear Steel Frame Analysis- Spearman Rank Order Correlation Coefficients between inputs 1	60
4.11	Results Set Nonlinear Steel Frame Analysis- Spearman Rank Order Correlation Coefficient between inputs 2	61
4.12	Result Set Nonlinear Steel Frame Analysis – Linear Correlation Coefficients between Input and Output Variables	61

**LIST OF FIGURE**

<b>Figure No.</b>	<b>Title</b>	<b>Page</b>
3.1	Steel Frame Model	12
3.2	Flow Chart of Methodology	14
3.3	Water Treatment Plant Shelter	15
3.4	Adding Element Type	19
3.5	Element Type	19
3.6	Define Geometric Properties	20
3.7	Adding Geometric Properties	20
3.8	Element Material Properties	21
3.9	Adding Element Material Properties	21
3.10	Mesh Size	22
3.11	Solution Controls	23
3.12	Deflection Contour Plot	24
4.1	Steel Frame Model	26
4.2	Input Random Variable Poisson's Ratio	29
4.3	Input Random Variable Density	30
4.4	Input Random Variable Young Modulus	31
4.5	Input Random Variable Force1 a	32
4.6	Input Random Variable UDL1 a	33

4.7	Result Set Nonlinear Steel Frame Analysis- Sampled Values for Output Parameter Max Deflection	38
4.8	Result Set Nonlinear Steel Frame Analysis- Mean Value History for Output Parameter Max Deflection	39
4.9	Result Set Nonlinear Steel Frame Analysis- Standard Deviation History for Output Parameter Max Deflection	40
4.10	Result Set Nonlinear Steel Frame Analysis- Minimum Value History for Output Parameter Max Deflection	41
4.11	Result Set Nonlinear Steel Frame Analysis- Maximum Value History for Output Parameter Max Deflection	42
4.12	Result Set Nonlinear Steel Frame Analysis- Histogram Of Input Variable Area	44
4.13	Result Set Nonlinear Steel Frame Analysis- Histogram Of Input Variable Poisson's Ratio	45
4.14	Result Set Nonlinear Steel Frame Analysis- Histogram Of Input Variable Density	46
4.15	Results Set Nonlinear Steel Frame Analysis- Histogram Of Input Variable Young Modulus	47
4.16	Result Set Nonlinear Steel Frame Analysis- Histogram Of Input Variable Force 1 A	48

4.17	Result Set Nonlinear Steel Frame Analysis- Histogram Of Input Variable UDL 1 A	49
4.18	Result Set Nonlinear Steel Frame Analysis- Histogram Of Output Parameter Max Deflection	50
4.19	Result Set Nonlinear Steel Frame Analysis- CDF of Input Variable Area	52
4.20	Result Set Nonlinear Steel Frame Analysis- CDF of Input Variable Poisson's Ratio	53
4.21	Result Set Nonlinear Steel Frame Analysis- CDF of Output Parameter Max Deflection	55
4.22	Result Set Nonlinear Steel Frame Analysis- Sensitivity Plot for Max Deflection	57

## CHAPTER 1

### INTRODUCTION

#### 1.1 Background of Study

Almost all oil and gas plants will have shelters to cater to the need of equipment, personnel or as an enclosure for production, storage. Shelter can be open, partially closed or fully closed. Shelter can be made of steel structures, concrete structure or even composite structures as well. This analysis study is focused to the fully closed shelter and made of steel structure. The shelter span length is 5 meter between column c/c. The high of the shelter is 3.5 meter for each floor. This shelter having two main frame structures which is the first building is two storeys and the other is three storeys. Frame steel structure have been analyzed for this study and element that were analyzed is beam. For shelter steel frame analysis included the forces (axial loads) and uniform distribution load (UDL).

This study is part of a comprehensive research that was conducted to investigate the structural behavior of such type of shelters. The study included the description and the results of a finite element analysis conducted on steel frame shelters. The analysis is carried out analysis covering various loading and geometry configurations. There are 8 variables used as input variables which are area, density, elasticity of modulus, Poisson's ratio, force and uniform distributed load. The smallest size shelter has a length  $L= 3.5$  m. This length can be doubled or tripled for larger shelters. The height of the section webs of the roof varies linearly following the roof profile and has a maximum value of 0.7 m at the middle of the roof.

The study have been described the finite element scheme and modeled of the shelter steel frame structure. The procedure to incorporate these properties into the finite element model is then described. Analysis, covering various loading and geometry conditions, is considered. Finally, the conclusions and result is made from the finite element analysis.

FEA is stand for finite element analysis is engineering software that is designed to accept input data and determine a beam design to meet the performance criteria. The FEA is also to be known as the finite element method which is FEM analysis is a numerical technique for finding approximate solutions of partial differential equations as well as of integral equations. The solution approach is based either on eliminating the differential equation completely for steady state problems, or rendering the equation into an approximating system of ordinary differential equations, which are then numerically integrated using standard techniques such as Euler's method and Runge-Kutta.

Frame is used all around us in many mechanical and structural engineering applications. They are commonly used to create a foundation or internal support for a larger structure, such as a building or a shelter. Frame analysis requires a combination of mechanical engineering, design principles, and material properties. The process typically involves factors including the frame shape, the material, and the design of the joints to allow one beam to be mechanically connected to other structural members.

For this project, the title is Nonlinear Finite Element Analysis of Steel Frame, it required to know and do an analysis using Finite Element Method (FEM) by using ANSYS software. Overall, this project involved in the analyzed and modeled steel frame shelter. After finishing and completing the whole project, student can get more skills in designing, analyzing and simulating using ANSYS software. To complete this study, first we need to improve the skill when using the related software so that it can help to get better results.

The introduction of software usage in the civil engineering industry has greatly reduced the complexities of different aspects in the analysis and design of projects, as well as reducing the amount of time necessary to complete the designs. Concurrently, this leads

to greater savings and reductions in costs. More complex projects that were almost impossible to work out several years ago are now easily solved with the use of computers. In order to stay at the pinnacle of any industry, one needs to keep at par with the latest technological advancements which accelerate work timeframes and accuracy without decreasing the reliability and efficiency of the results.

## **1.2 Problem Statement**

In analysis and design stage for structure, the phase now had become unstable which is many failure of structure had happen because of many factors. From this problems comes an idea to make a finite element analysis to one of the structure and steel frame of shelter is modeled to understand it behavior to avoid it fail. For this study a three storey steel frame shelter is modeled. The selected beam form the steel frame shelter will be analyzed to find the objectives of this study. The boundary condition for that beam is simply supported. The criteria of beam to be selected for analyzed process based on the critical condition of that beam itself, beam will be chosen based on the maximum load.

## **1.3 Objectives**

The objectives for this study are:

- a. to analyze the maximum deflection of nonlinear steel frame using finite element method
- b. to determine the behavior of steel frame structure

## **1.4 Scope of Study**

This study emphasis only on the steel frame structure and nonlinear steel frame analysis of the shelter. Analysis is just a method performed to determine the maximum deflection for the steel frame structure since there a lack of previous researched and study on the nonlinear behavior of steel frame shelter. However, in order to make the project run

smoothly, scopes have been determined to guide and to prevent any misleading. Scopes of this project are as below:

- a. Finite Element Analysis (FEA) Software
- b. Analysis process for steel frame shelter
- c. Selection of an available steel frame will be used in this project
- d. Boundary condition of the steel frame
- e. Simulation and analyzing process of steel frame using ANSYS

#### **1.4.1 Finite Element Analysis (FEA) Software**

Finite element analysis for steel frame shelter used software (ANSYS) to simulate and later get the result. After that the analysis were run and all the data and variables are determined. Finite element analysis is a numerical method which provides solutions to problems that would otherwise be difficult to obtain. Because the result would be difficult to obtain so the analysis were solved by using software.

#### **1.4.2 Analysis Process for Steel Frame Shelter**

The analysis phase for this study is to get the maximum deflection value for steel frame structure. Either Spearman rank order correlation coefficients or Pearson linear correlation coefficients are used to get the data. The details of analysis process are explained on chapter 3.

#### **1.4.3 Selection of an Available Steel Frame will be used in this project**

Selection of steel frame for this study is important part as guiding the steps need to be carrying out during this study in order to achieve the research objective as stated . To get all data of the steel frame many sources are used as references such as BS 5950: Part 1: 2000 to get the information for load applied. The critical condition of steel frame will be considered.



#### **1.4.4 Boundary condition of steel frame**

For this study the boundaries condition is simply supported so the pinned and roller support is used. Boundary condition is important to make sure the steel frame is in stable condition to avoid it fail and to make sure the analysis can be done. It's important because the boundary conditions are known to have a major role on the structural behavior.

#### **1.4.5 Simulation and analyzing process of steel beam using ANSYS**

The simulation is all the steps that been done using ANSYS to get the result. The analysis is run after all the data and variables are determined. This study use Monte Carlo simulation methods. After all the maximum deflection value is determined by analyzed all the data than the conclusion can be made based on the objective of the study as stated before.

### **1.5 Significant of Study**

When study is complete more information about nonlinear behavior will get from the finite element analysis. The maximum deflection for steel frame analysis also will be determined at the end of the study. From this study the more information about the steel frame element before we can design a shelter can be used after the analyzing process done. This study will discover that neglecting nonlinear effects can lead to serious design errors. Furthermore this study will give a view how nonlinear analysis can help in avoiding overdesign and build better structure. To complete this project, first student need to improve the skill when using the related software and also in operating and handling the welding machine so that it can help the student better. The application of simulation software such as Ansys can help us understand better about nonlinear finite element analysis.

## CHAPTER 2

### LITERATURE REVIEW

#### 2.1 Introduction

Hence the concept of a shelter was born. The first publication of the concept appeared in Civil Engineering magazine in 1974. Because of limited resources and manpower, development of the residential shelter concept continued at a slow pace for a number of years, and few people outside the research community were aware of the work being done. The program documented the total destruction of the residential subdivision outside and then featured photos of debris impacts on shelter components and illuminated the concept of the aboveground storm shelter. The Federal Emergency Management Agency (FEMA) further popularized the concept by publishing a booklet, FEMA 320: Taking shelter from the storm: Building a safe room inside your house Federal Emergency Management Agency 1999. (Kiesling and Goolsby, 1974)

Formulation of FEMA 320 shelter design was based on several factors. First, the shelter had to remain intact during the extreme wind event taken as a 112 m/ s. This requirement takes into account performance of the basic shelter structure walls, roof, attachment to foundation, and door. Second, the shelter had to resist overturning. This is a function of the competency of the attachment of the shelter to its foundation. Third, the walls and door had to resist design missile debris impact, protecting the occupants within. A brief review of previous studies on the application of the finite element method to the analysis of shelter is in this chapter. This chapter will discuss on some literatures that give

information about finite element analysis and how the nonlinear analysis of shelter process will be done.

Every structure that had been invented by the engineer had their own specifications that have to be followed. Steel frame also does not exceptional. There are a lot of types of beams that we can found in some of the structure depending on the use of the structure itself. 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 element lay in the application of such techniques for structurally related problems. (Pepper, D.W and Heinrich J.C, 2006)

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 codes development, and may find the code sources available in such texts as that by Zienkiewicz<sup>2</sup> to be 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)

## **2.2 Shelter Structural Analysis Using ANSYS**

The complexity of the structure and loading of a storm shelter are clearly not amenable to classical, closed-form solution numerical methods specifically finite-element analysis (FEA) are required. In the past 20 years, finite-element analysis has found widespread use for the analysis of structural systems under extreme loading, such as seismic, blast, and wind. Numerous commercial software packages are available that facilitate the use of FEA techniques. These programs provide streamlined procedures for prescribing nodal point locations, element types and locations, boundary constraints, steady and or time-dependent load distributions, and the specific solution information desired. Several of these packages notably ALGOR, ANSYS, DIANA, and ABAQUS, are

sufficiently sophisticated to have been accepted as basic research tools. (Adams and Azkenazi, 1998)

ANSYS was chosen for this project because it has a wide range of utilities that allow the modeling and analysis of the shelter to be performed quickly and efficiently. Changes in the model and loading can be quickly implemented. The shelter model can be created in one of several commercially available computer-aided design packages and imported into ANSYS. The shelter's surfaces can be defined within the original model, and these are preserved by ANSYS. Mesh generation is automatic, and several different types of elements can be specified. Constitutive models can be chosen from ANSYS's built-in library, or they may be user-defined. All types of connections between elements, including elements, can be defined. This is particularly useful in modeling composite members such as the FEMA 320 steel shelter, which has a frame covered and steel sheeting.

### **2.3 Nonlinear FEA 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.

The finite element model used is based on a consistent shell element that was initially developed by Koziey and Mirza. This model was extended to non-linear behavior by El Damatty et al and then further modified to simulate the behavior of shelter by Mikhail and El Damatty. The main advantage of the consistent shell element is being free from the spurious shear modes associated with isoperimetric shell elements. The element has a triangular shape and consists of 13 nodes. The degrees of freedom of the shell element include global displacements  $u$ ,  $v$  and  $w$  directed along the global axes  $x$ ,  $y$  and  $z$  and two through thickness rotational degrees of freedom  $a$  and  $b$ . (Koziey and Mirza, 1994)

In 1943, R. Courant became the first person who developed FEA when he was managed to utilize the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems. FEA was limited use by early 70's to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Later, FEA has been developed to an incredible precision since the rapid decline in the cost of computers and the phenomenal increase in computing power. Supercomputer nowadays seems to calculate and produce accurate data for all parameters. (R.Courant, 1943)

In the finite element model, each individual triple wall panel was simulated using a number of triangle shell elements. A typical finite element mesh for the outside and the inside surfaces of the shelter, respectively. The sections were simulated as flat vertical panels using a number of triangular shell elements that connect the inner and outer triple wall panels. Due to the symmetry of the shelter in the longitudinal direction, only one half of the shelter was modeled. The length of the model was considered to be equal to  $L/2$ . A total number of 380 elements were used to model one half of a shelter. The bases of the side walls and the end beam were assumed to be fixed. A continuity of displacements and rotations was imposed at the interface between the side walls and the roof.

This study is part of a comprehensive research program that was conducted at the University of Western Ontario to investigate the structural behavior of such type of shelters under severe wind conditions. The paper includes the description and the results of a finite element analysis conducted on full-scale cardboard shelters. The analysis is carried out using an in-house developed laminated shell. Seven cases of analysis covering various loading and geometry configurations, including the effect of door and window openings, were conducted. The loads considered in the analyses are those due to the non-typhoon wind pressure. It can be seen that the length  $L$  is a variable parameter. The smallest size shelter has a length  $L = (3.65 \text{ m})$ . This length can be doubled or tripled for larger shelters. The mechanical properties that are incorporated in the finite element model are based on the test results conducted on the triple wall corrugated cardboard components at the University of Western Ontario.

Two types of forces were used to represent wind-induced pressure: uniform surface load and nodal force. Surface pressure represented the uniform distributed wind loads on the sides and roof elements. Nodal forces were used to represent the load transferred from the door to the shelter by the door frame anchors to the walls. Windward wall pressures the shelter and therefore are not visible. The loads associated with different wind speeds are determined with coefficients and factors specified by FEMA 361. Linear stress analysis was used. Element specifications were defined. The mass density and the modulus of elasticity of the elements were required to perform this type of analysis. For the shelter, isotropic plate material was chosen for all element types. The modulus of elasticity was 9.31 (103 Mpa) 1.35 (106 psi).

The paper describes the finite element scheme and the modeling of the shelter structure. The material properties that are obtained from the component tests are briefly outlined. The procedure to incorporate these properties into the laminated orthotropic finite element model is then described. Seven cases of analysis, covering various loading and geometry conditions, are considered. Strains resulting from the finite element analyses are compared with the ultimate strains of the material, obtained from the component tests, in order to assess the level of safety provided by the shelters. Finally, the conclusions drawn from the finite element analysis are given.

Deformations resulting from the finite element analysis were evaluated and then plotted along various cross-sections. In this paper, deformations are presented along transverse and longitudinal cross-sections. Both cross sections are located in a plan that passes through the central point of the shelter. The strains resulting from the analysis are given along two local axes  $x$  and  $y$  which are located in the plane of the shelter components. For the side walls and the roof, the  $y$  axis is directed along the longitudinal direction (the global  $y$ -axis), while the  $x$  axis is perpendicular to  $y$  axis.

## CHAPTER 3

### METHODOLOGY

#### 3.1 Introduction

Methodology is important part as guiding the steps need to be carrying out during this study in order to achieve the research objective as stated in chapter 1. Several steps are taken to conduct this research accordingly. The steps are:

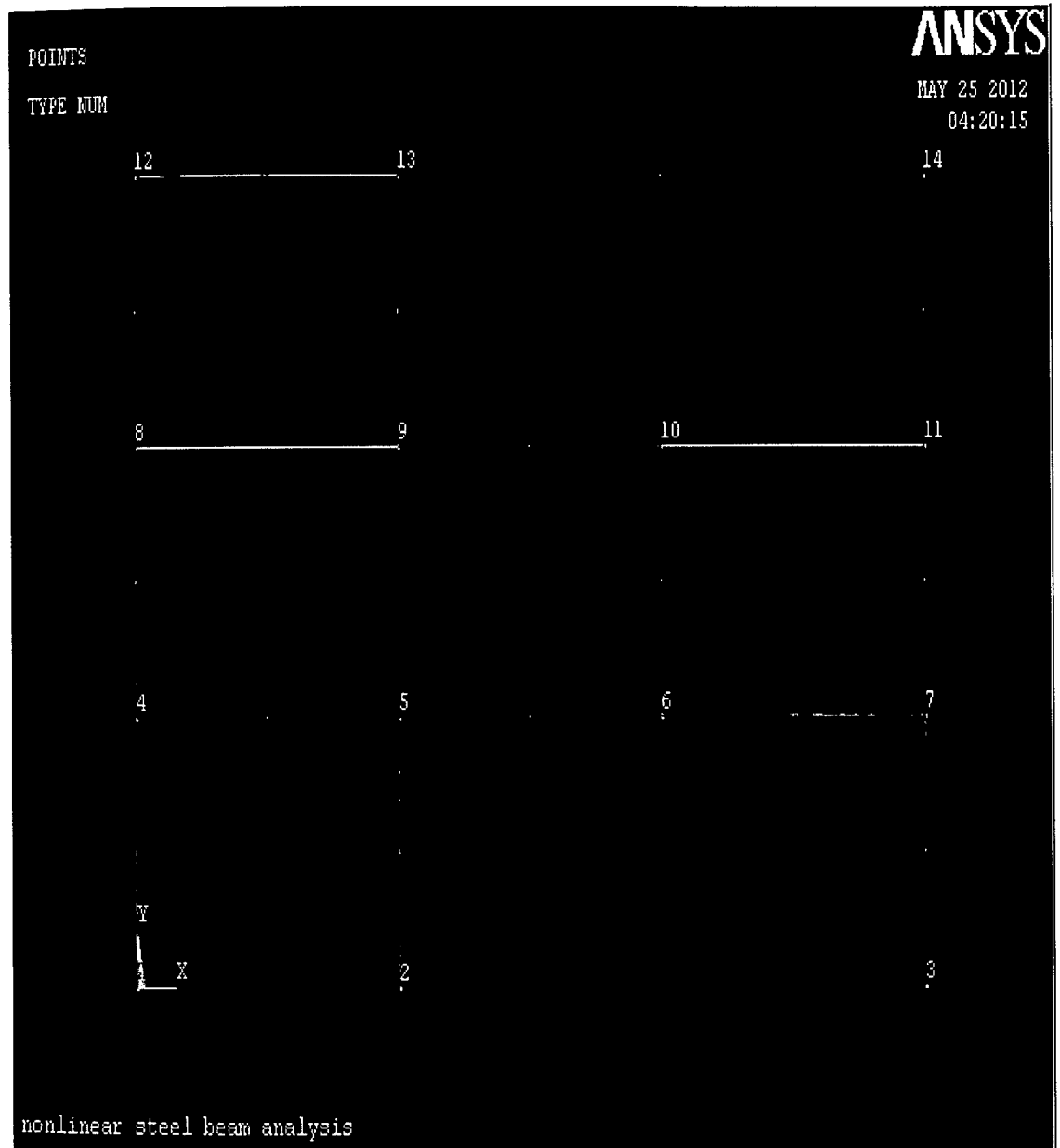
- i. get all data of the selected structure (steel frame)
- ii. modeling the structure (steel frame)
- iii. simulation the model using the software
- iv. analyzed the data
- v. makes conclusion based on researched been made

##### 3.1.1 Get all data of the selected structure (steel frame)

Methodology is important part as guiding the steps need to be carrying out during this study in order to achieve the research objective as stated in chapter 1. To get all data of the steel frame many sources are used as references such as BS 5950: Part 1: 2000 to get the information for load applied. After get all data we can proceed to the next step.

### 3.1.2 Modeling the structure (steel frame)

Steel frame structure was modeled using software (ANSYS). The model of steel frame structure is shown in figure:



**Figure 3.1:** Steel frame model