PROBABILISTICS FINITE ELEMENT ANALYSIS OF 3D PORTAL FRAME USING ANSYS

TEO KAH WEI

B.ENG (HONS.) CIVIL ENGINEERING

UNIVERSITY MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

DECLARATION OF THESIS AND COPYRIGHT

Author's full name   :   TEO KAH WEI

Date of birth   :   19 APRIL 1991

Title   :   PROBABILISTICS FINITE ELEMENT ANALYSIS FOR 3D

                                PORTAL FRAME USING ANSYS

Academic Session   :   2014/2015

I declare that this thesis is classified as:

☐   **CONFIDENTIAL**   (Contains confidential information under the Official Secret Act 1972)*

☐   **RESTRICTED**   (Contains restricted information as specified by the organization where research was done)*

☑   **OPEN ACCESS**   I agree that my thesis to be published as online open access (Full text)

I acknowledge that Universiti Malaysia Pahang reserve the right as follows:

1. The Thesis is the Property of University Malaysia Pahang
2. The Library of University Malaysia Pahang has the right to make copies for the purpose of research only.
3. The Library has the right to make copies of the thesis for academic exchange.

Certified By:

(Student's Signature)	(Signature of Supervisor)
910419-06-5739	DR. CHENG HOCK TIAN
New IC / Passport Number	Name of Supervisor
Date :	Date :

NOTES: *If the thesis is CONFIDENTIAL or RESTRICTED, please attach with the letter from the organization with period and reasons for confidentiality or restriction.

PROBABILISTICS FINITE ELEMENT ANALYSIS OF 3D PORTAL FRAME
USING ANSYS

TEO KAH WEI

Report submitted in partial fulfillment of the requirements

for the award of degree of

B.Eng. (Hons.) Civil Engineering

Faculty of Civil Engineering & Earth Resources

UNIVERSITI MALAYSIA PAHANG

JUNE 2015

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Civil Engineering (Hons.).

Signature   :

Name of Supervisor   : DR. CHENG HOCK TIAN

Position   : Undergraduate Research Project Supervisor

Date   :

STUDENT'S DECLARATION

I hearby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature   :

Name   : TEO KAH WEI

ID Number   : AA 12009

Date   :

This thesis is dedicated to my beloved family and partner.

ACKNOWLEDGEMENT

First of all, I am truly feel appreciate and thankful to my Supervisor, Dr. Cheng Hock Tian. He had guided and helped me a lot in my research and also thesis writing. He always teaches me very patiently throughout the process of analysis and gives me suggestion when I face difficulties. In order to enrich my knowledge, he encouraged me to read more by providing a lot of examples for me.

Besides that, I would like to thanks to the research team that had gave a lot of suggestions to improve my shortage. Furthermore, I would like to thanks my parents also for their encouragement, support and understanding to me when I worked hard in this final year project of my degree course. I feel deeply sorry to them as I had neglected them throughout this research is running. Other than that, I would like thanks the most important person in my life, Shirly Chow Wannyen who gave a lot of support to me. Without her support, I would not complete this project smoothly. Once again, thank you to her.

I am grateful to thank UNIVERISITY MALAYSI PAHANG that provided me a chance to complete my study and final year project. I also want to thank you to my course mates and seniors that helped me a lot in using ANSYS software, sharing knowledge to me. Just to let them know, I appreciate all of it.

Lastly, I feel happy to complete this research as I had used a lot of time in completing this research. The hard work is paid off after I see the results come out is satisfied. Thank you very much.

ABSTRACT

In this research, a 3D portal frame was analyzed by manual calculation and also ANSYS program with the probabilistic method. This analysis is to determine the stress, strain, deflection and deformation of the structure. Besides that, the structure is checked accordingly to Eurocode 3 also. Nowadays portal frame is widely used for industrial building. It is very stable in term of everything and low in cost. A 500N/mm^2 of yield strength was choose for the 3D portal frame. The material and geometry of the space truss was satisfied for this research since it passed all the design criteria in designing for tensile, compression and also buckling. The numerical results show very good agreement with manual calculation results. From the results of simulation, we get to know the real behavior of the structure under the applied loads. In probabilistic analysis, we get the results of cumulative distribution function, histogram plot, sensitivity plot and simple history plot by 10000 times of simulation for any input and output.

ABSTRAK

Dalam kajian ini, kerangka portal 3D dianalisis dengan pengiraan manual dan juga program ANSYS dengan kaedah kebarangkalian. Analisis ini adalah untuk menentukan tekanan, terikan, pesongan dan kecacatan bentuk. Selain itu, struktur diperiksa dengan sewajarnya kepada Eurocode 3 juga. Pada masa kini kerangka portal digunakan secara meluas untuk bangunan industri. Ia adalah sangat stabil dari segi segala-galanya dan rendah kos. Kekuatan alah $500N/mm^2$ adalah pilihan untuk kerangka portal 3D. Bahan dan geometri kekuda ruang yang berpuas hati untuk kajian ini kerana ia lulus semua kriteria reka bentuk dalam mereka bentuk untuk tegangan, mampatan dan juga lengkokan. Keputusan berangka menunjukkan perjanjian yang sangat baik dengan keputusan pengiraan manual. Dari hasil simulasi, kita mengenali tingkah laku sebenar struktur di bawah beban yang dikenakan. Dalam analisis kebarangkalian, kita akan mendapat hasil fungsi kumulatif pengedaran, histogram plot, plot sensitiviti dan sejarah plot mudah oleh 10.000 kali simulasi dari segi input dan output.

TABLE OF CONTENTS

CHAPTER 1   INTRODUCTION

CHAPTER 2   LITERATURE REVIEW

CHAPTER 3   METHODOLOGY

CHAPTER 4   RESULTS AND DATA ANALYSIS

CHAPTER 5   **CONCLUSIONS AND RECOMMENDATIONS**

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

tw   Web thickness

tf   Flange thickness

hw   Height of web

i   Height

b   Width

d   Depth

A   Area of section

I   Moment of inertia

W_{pl} Plastic modulus

i   Radius of gyration

N   Axial load

V   Shear force

M   Moment

α   Imperfection factor

γM0   Partial factor for resistance of cross-sections whatever the class is

γM1   Partial factor for resistance of members to instability assessed by member checks

λ   Slenderness value

Ø   Value to determine the reduction factor

X   Reduction factor

L_{cr} Buckling Length

K_{zy} Interaction factor

[K]   Global-coordinate structure stiffness matrix

LIST OF ABBREVIATIONS

2D	Two Dimensional
3D	Three Dimensional
CIVIFEM	Civil Finite Element Method
LatBuck	Lateral Buckling
ChckAxis	Check Axis
BMSHPRO	Beam and Shell Properties
CS	Coordinate System
LS	Load Step
DOF	Degree of Freedom
PRES	Pressure
GAUS	Gaussian
DENS	Density
ELASTIC	Elastic modulus
POISON	Poison ratio
LOAD	Point load
WINDLOAD	Wind load
TEMP	Temperature
PDF	Probabilistic density function
CDF	Cumulative distribution function
MAXIMUMDEFLECTION /MAX_DEFLECTION	Maximum Deflection

CHAPTER 1

INTRODUCTION

1.1   INTRODUCTION

There are many types of frame in steel construction industry. Portal frame is one of the famous frame that is widely used especially for single-storey industrial building. The dominance of the steel construction for single-storey non-domestic buildings is demonstrated when the estimation of usage of steel frames by non-domestic buildings is more than 90% (Davidson & W. Owens, 2012). Steel is more preferred to be used as material for portal frame. There are some characteristics of steel that allow it to be chosen as material for portal frames. They are relatively light, long-span and durable. Besides that, they are very simple, can be quickly erected, and developments in steel cladding have enabled architects to design economical buildings with attractive appearance to meet a wide range of applications and budgets. Portal frame has great stability against lateral and vertical actions. This make portal frame the best choice of single-storey building. A portal frame can be very rigid and stable in plane with moment-resisting connections (Davidson & W. Owens, 2012).

There are some advantages of using portal frames. Portal frames can span 15-45 meters easily without support. This will allow larger area of free floor space to be used which is very suitable for single-storey industrial building. Erection of portal frames can be 20% quicker than common building. Besides that it is more economical because metal

in portal frame is lesser than a truss frame. The cost also can be saved by using universal beams to construct (University Deakin, 2004).

The 3D portal frame that going to be analyzed in this research is to be 12m in total height. The width of the portal frame is 12m also as shown in Figure 1.1 below. As the frame has 12m in height, it is sustaining wind load. Wind load is considered in this research.
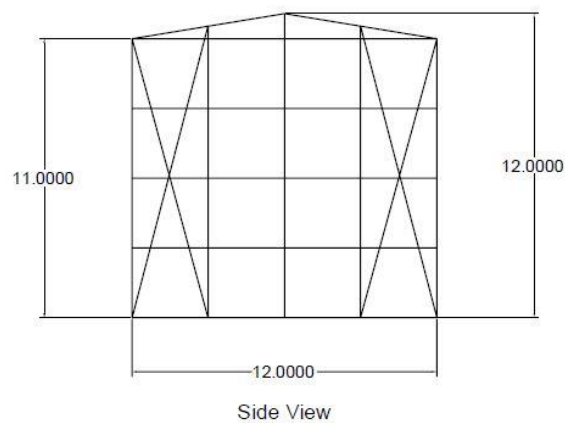
Figure 1.1: Side View and Dimension of 3D Portal Frame

The portal frame is offset with 5m distance until 40m to form an single-storey industrial building. According to Figure 1.2 below, there are extra bracing at the top middle of the roof because there are opening because it is the entrance of the building.
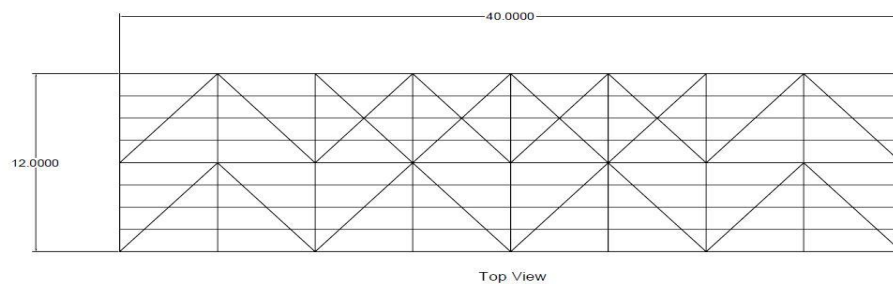
Figure 1.2: Top View and Dimension of 3D Portal Frame

1.2   PROBLEM STATEMENT

2D Portal frames are easy to be analyzed manually. However, it is hard to analyze 3D portal frame. There are a few layer portal frames connecting by bracing to form 3D portal frames. The structure is complicated. It also takes a lot of time to analyze it which also cause higher cost to design a structure with 3D portal frame. There are a lot of software can be used to analyze portal frame. ANSYS is one of the software that very useful in analyzing complicated structure. Modeling 3D structure is also very easy by using ANSYS. By using ANSYS, it helps to save a lot of time and cost when designing a structure.

Although there are much research on truss, beam, column, frame and bridge outside, but they are mostly on 2D analysis and not applicable when it comes to the real construction of it. There is less research on 3D portal frame using ANSYS program also due to unfamiliar using of ANSYS to analyze 3D structure. In this research, maximum deflection, deformed shape, and checking according to Eurocode 3.

1.3   OBJECTIVE

Research objectives are an important part for every project or thesis in order to conduct the research well. It acts as guidance for researcher to achieve the final objective and avoid researcher to digress. Objectives are also important for researcher to understand and remind themselves of the criteria and testing that should be done in a research. The main objectives of this research are:

   (i)   To determine the behavior of 3D portal frame
   (ii)   To make sure the portal frame pass all the code checking
   (iii) To make sure the portal frame is stable against deflection and deformation
   (iv)   To determine the response behavior of portal frame under deflection
   (v)   To determine the behavior of portal frame under point load and wind load

1.4   SCOPE OF STUDY

This research will be carried out by using ANSYS to analysis 3D portal frame. Modeling is to be done in ANSYS. The load is assumed to be point loads only because cladding is calculated and transferred to node as point loads. There are only primary and secondary steel works included which are portal frames, bracing, purlins and side rails. Wall cladding is assumed to be calculated and transferred to node as point loads. The point loads are to be assumed to include permanent and variable actions. In the meanwhile, wind load is only applied on one side of the structure only. Wind load is also applied at the main steel works only which is portal frame. Lastly, in post-processing step, we will only check the deflection, deformed shape, and against Eurocode 3 checking.

1.5   EXPECTED OUTCOME

This research claims to find out the behaviors and passing code checking of 3D portal frame. The behaviors are deformation, deflection, tension checking, compression checking and lateral torsional buckling checking according to Eurocode 3.

1.6   SIGNIFICANCE OF STUDY

This research will be a significance endeavor in providing the information about the steps and results of analyzing 3D portal frame by using ANSYS. Maximum deformation, deflection axial stress and strain are able to be determined. It is really good to practice to analyze the portal frame and make sure it is safe before constructing it. Furthermore, this research can help to save the cost of the construction by determining the optimum dimension of portal frame should be used. It is really important to learn and design steel structure in Malaysia as a lot of advance country is using portal frame for the single-storey buildings. This definitely will enhance the reputation of Malaysia with advance engineering skills. In a nutshell, this study is very significant to Malaysian and the field of civil engineering.

CHAPTER 2

LITERATURE REVIEW

2.1 PORTAL FRAME

2.1.1 Introduction to Portal Frame

Portal frames are structures which consist of column, pitched or horizontal rafters. They are connected to each other by moment resisting connections. The rigidity of connections and the bending stiffness of the members are very advantageous on providing the frame to resist lateral and vertical actions. Bending stiffness of the members can be increased by providing a suitable haunch or deepening of the rafter sections. The frame structure is continuous and stable in plane. It can provide clear span by bracing (Steel Construction, 2014). According to estimation, portal frames are the most common form from 50% of hot-rolled steel that fabricated into single-storey buildings (Duoc, et al., 2013).

2.1.2 Anatomy of Typical Portal Frame

A portal framed building is made up of a series of transverse frames which braced longitudinally. The main steelwork that form portal frames are column and rafters. The end frame is usually a portal frame also. The light gauge secondary steelwork are

important to link up all the frames together to form a building. They usually consists of side rails for the wall and purlins for the roof. They helps to support the building envelope and restraining primary steelwork. After that, the building is enclosed by roof and wall cladding which functioning to separate the enclosed space from the external environment. Some of the cladding will provide thermal and acoustic insulation. Besides that, cladding is playing role to transfer loads to secondary steelwork. It also function to restrain the flange or rail that attached to it (Steel Construction, 2014).
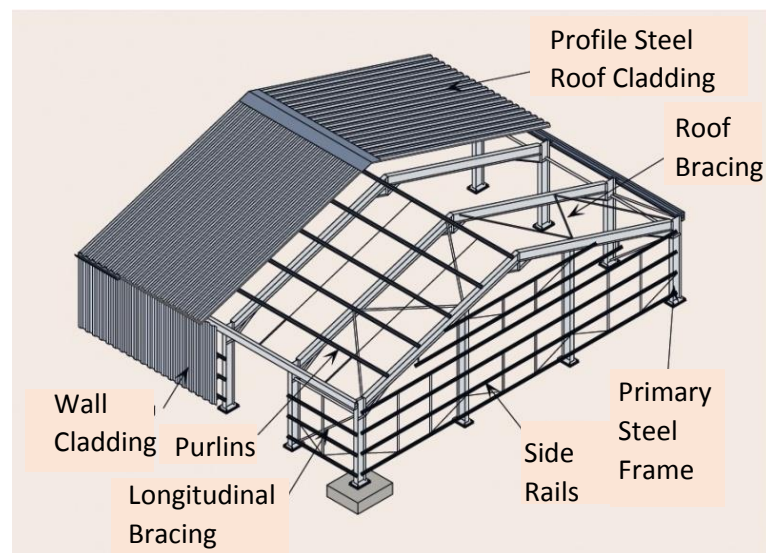
Figure 2.1: Principal components of a portal framed building (Steel Construction, 2014)
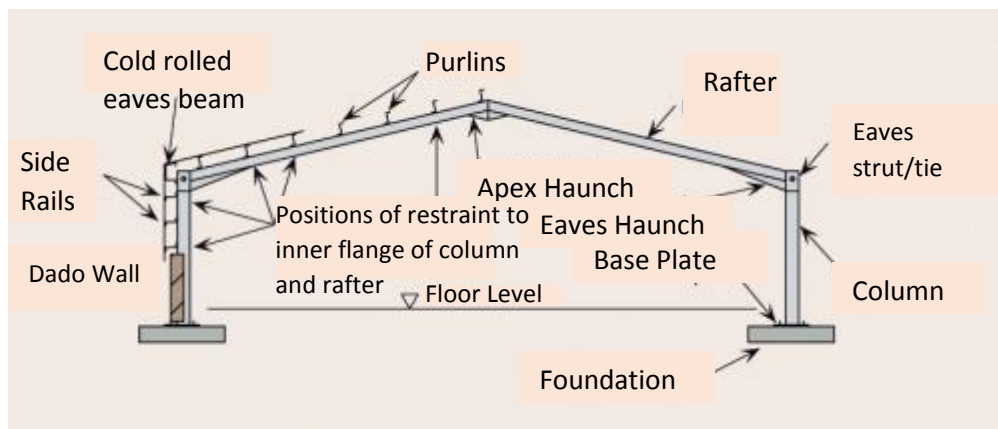
Figure 2.2: Cross Section showing a portal frame and its restraints (Steel Construction, 2014)

2.1.3 Types of Portal Frame

There are many types of portal frames and they are constructed according to the function of the building.
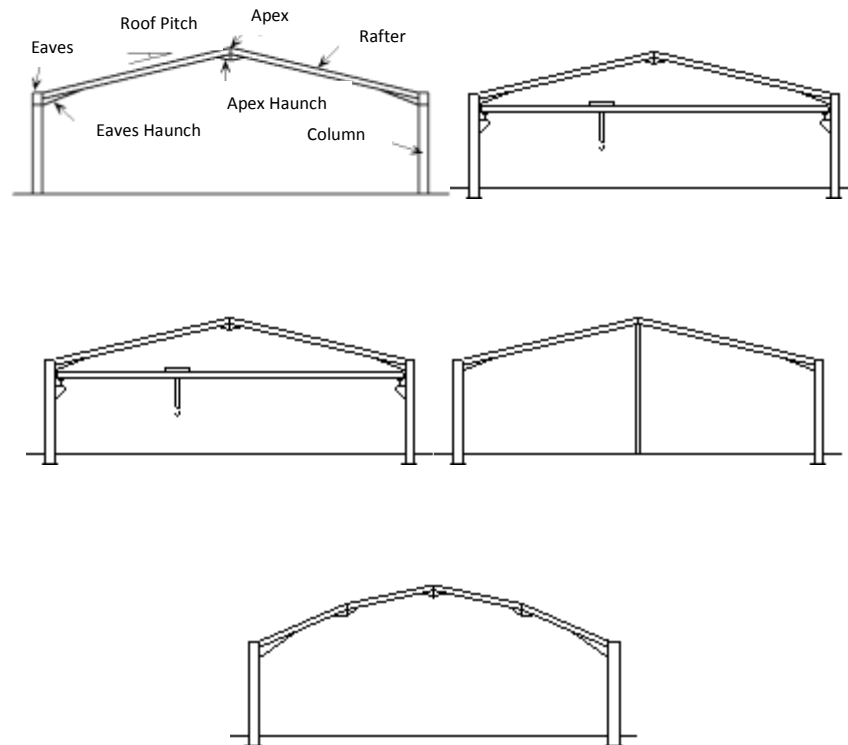
Figure 2.3: Types of Portal Frames (Steel Construction, 2014)

2.1.4 Frame Analysis At Ultimate Limit State (ULS)

2.1.4.1 Plastic Analysis

Most of the economical frames are analyzed by plastic analysis. This is because it usually allows relatively large redistribution of bending moments throughtout the frame due to plastic hinge rotations. The plastic hinge rotation usually occur at sections where the bending moment reaches the plastic moment or resistance of the cross section at loads below the full Ultimate Limit State (Steel Construction, 2014). Most efficient designs in

the majority of cases are usually produced by plastic design nowadays (Ross, James, Tiku, Duoc, & Wei Sha, 2014).
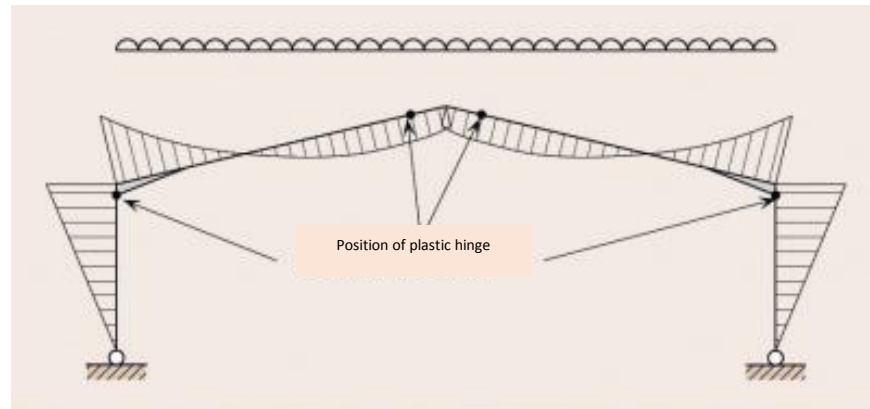
Figure 2.4: Bending Moment Diagram Resulting from the Plastic Analysis of A Symmetrical Portal Frame under Symmetrical Loading (Steel Construction, 2014)

2.1.4.2   Elastic Analysis

Elastic design is usually used when frame design is controlled by serviceability limit state deflections (Ross, James, Tiku, Duoc, & Wei Sha, 2014). Elastic analysis is usually done by using finite element program ANSYS (Duoc, et al., 2013).

2.1.5   Actions

2.1.5.1   Permanent Actions

Permanent actions are the total self-weight of structural and non-structural members that are taken into considerations when designing a structure. Besides that, the coatings of the structure after execution should be taken into consideration also (BS EN

1991-1-1:, 2002). For steel structure, permanent actions are the self-weight of the steelwork and cladding. They can be calculated from their own unit weight which can be obtained from manufacturer or Eurocode (Steel Construction, 2014).

2.1.5.2   Variable Actions

2.1.5.2.1.   Roof Loads

Loads on roofs are given in Eurocode and they are depend on the roof slope. The roof loads for local checking of roof materials and fixings should be applied vertically (Steel Construction, 2014). Loads on roofs should not be combined with either snow or wind loads (BS EN 1991-1-1:, 2002).

2.1.5.2.2   Wind Actions

Wind actions are extremely complex. They can influence the final design of most of the buildings (Steel Construction, 2014). For the whole time, it is hard and to determine the wind pressure coefficient for the complex structure. Eurocode gave the chance to analysis the structure with the presence of wind loads. Wind action is included in Eurocode (M.T. & Jose, 2014).

2.1.6   Connections

Connections are the key point of moment resisting for the portal frame structure. The major connections in a portal frame are the eaves and apex connections. Both of the connections are working together to sustain loads. They could be very important during design. Therefore, eaves connections must able to carry a very large bending moment. Connections can be arranged to reduce the cost by minimizing the use of additional

reinforcement such as stiffeners. There are several ways that can minimize the cost by applying the benefits of connections effectively. Haunch can be deeper, extending the eaves connection above the top flange of the rafter and choosing a tougher column section (Steel Construction, 2014).
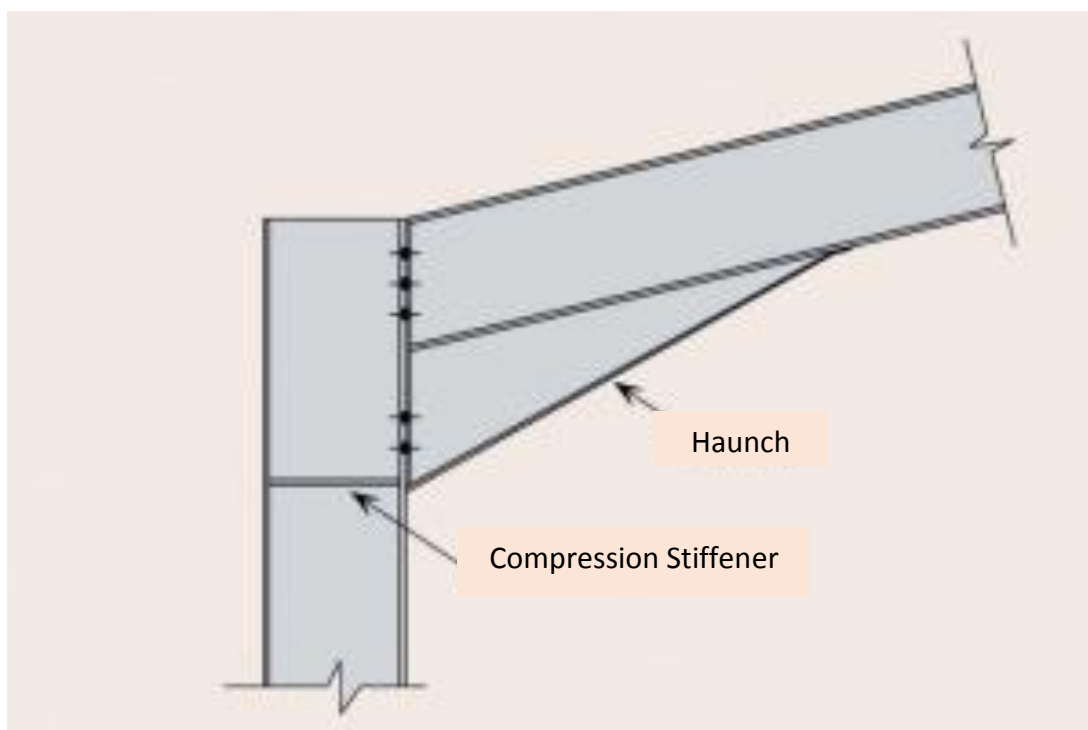
Figure 2.5: Eaves Connections
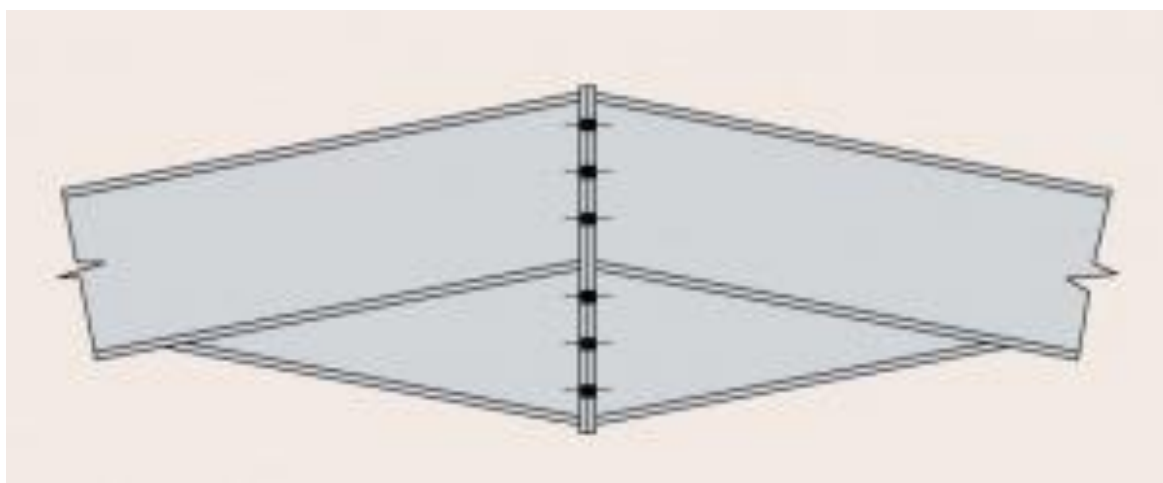
Figure 2.6: Apex Connections

2.2   ANSYS PROGRAM

2.2.1   Introduction

ANSYS is a very useful software in finite element modeling to numerically solve a wide variety of mechanical problem. This includes both linear and non-linear static or dynamic structural analysis, heat transfer and fluid problems, as well as acoustic and electromagnetic problems (University of Alberta - ANSYS Tutorials, 2001)

To analysis a structure, there are three stages need to be done in order to run the analysis. The first stage is Preprocessing stage which modeling must be completed in this stage. In order to complete the model of the structure, material, type of cross section, element type, beam properties and element type must be defined. Nodes and elements will completed to form the model of structure in this stage also.

Second stage is the solution stage. In this stage, the type of support of the structure will be determined and defined. Loads will be applied in this stage also. Third stage is the post-processing stage. This stage is very important to generate the results of the analysis. There are several types of checking can be done in ANSYS including Eurocode.

2.2.2   Portal Frame ANSYS Analysis

ANSYS is a powerful tool that able to analysis the structure thoroughly from the whole structure of the portal frame until the detail analysis of beam, column and lastly connection. In the meanwhile, it able to generate the simulation of the portal frame under various type of condition including the analysis of portal frame under fire conditions. ANSYS is a non-linear finite analysis tool (P.J., R.P., M.W. , & A.H. , 2008). ANSYS is also capable of presenting the actual behavior of the connections of the steel portal frame

structure (Elsayed, Mohamed, Hamdy, & Mohamed, 2010). Figure 2.7 shows the ANSYS modeling of the connection of the portal frame.
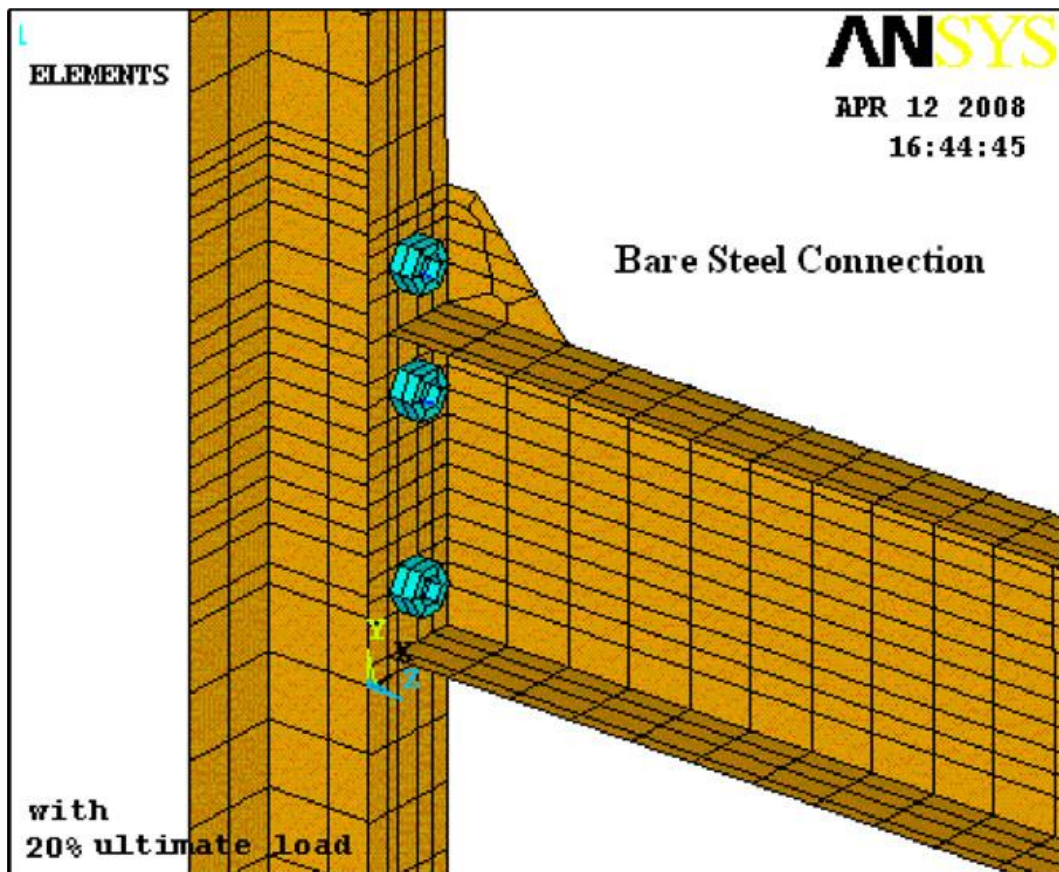
Figure 2.7: ANSYS modeling of the connection of the portal frame

2.3   ANALYSIS METHOD

2.3.1   Finite Element Analysis

Finite element method is the method to analysis the entire structure by simplify and calculate it separately. This method gives more accurate estimate of the stresses in the members. Thus, facilitating optimum design. Finite element method breaks the given component into small elements which all the elements are connected at nodes. Finite

element method is an extension of Rayleigh-Ritz method. This method helps to solve the difficulty to deal with large and complex structure (G Lackshmi, 2009)

There are two types of finite element analysis (G Lackshmi, 2009):

(i)   Thermal analysis – This analysis deals with linear operations which is steady state or transient heat transfer by conduction and convection. Besides that, it also deal with non-linear operation which is radiation. It also able to estimate the temperature distribution in the component. These analysis can form another load condition to solve structural analysis.

(ii)   Structural analysis – Finite Element able to estimate stresses and displacements for a structure. It also able to analysis structure under various type of loads such as gravity, wind, pressure and temperature. Other than that, dynamic load may also be considered.

Frame finite element analysis is usually analyzed using nonlinear theory by combining Shanley's tangent method and incremental Newton-Raphson equations. Finite element analysis capable to solve inelastic frame stability problems for 2D frames (Mohamed, David, & M., 1983). Figure 2.8 shows the effect to deflections when different type of loads are applied on frame using finite element method.
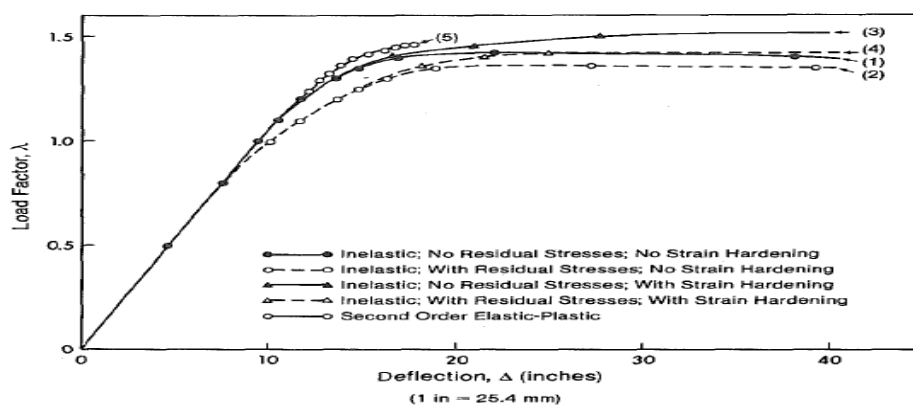
Figure 2.8: Effect to deflections when different type of loads are applied on frame using finite element method (Mohamed, David, & M., 1983)

CHAPTER 3

METHODOLOGY

3.1   INTRODUCTION

The methodology of the research is to be discussed in this chapter thoroughly by showing the step of analyzing single-storey industrial building formed by portal frame. This research is mainly about the finite element analysis and code checking by Eurocode 3 of 3D portal frame. In this research, ANSYS with integration of CIVILFEM computer program was used to analyze the structure of 3D portal frame. This program is very cost saving with the function to do simulation experiment. The analysis of the structure failure also able to be analyzed in this program. The material, cross section and element type of the steel structure are all provided in the program as it is integrated as CIVILFEM. The specification of this simulation is shown in Table 3.1 below.

Table 3.1: Specification of 3D portal frame

Specification	Used
Element Type	3D Lin. Beam 188
Shear Modulus	81E9 Pa/ 81E3 N/mm^2
Young's Modulus of Material	210E9 Pa/ 210E3 N/mm^2

The analysis of the 3D portal frame can be mainly divided into three steps. The steps are preprocessing process, solution process and post processing process. In preprocessing process which is the first process, the model of 3D portal frame was created by specifying the material, cross section, element type and then creating its nodes and elements to form the 3D portal frame. The step is followed by solution phase which the type of support of the structure is to be used for the analysis. Besides that, loads are going to be put on the nodes of the structure also in this step. The 3D portal frame structure was analyzed and solved at this stage. Lastly, the post processing process will be the stage to get the results that stated in the objective. The process of this research from the starting until the end was shown in Figure 3.1 as flow chart.
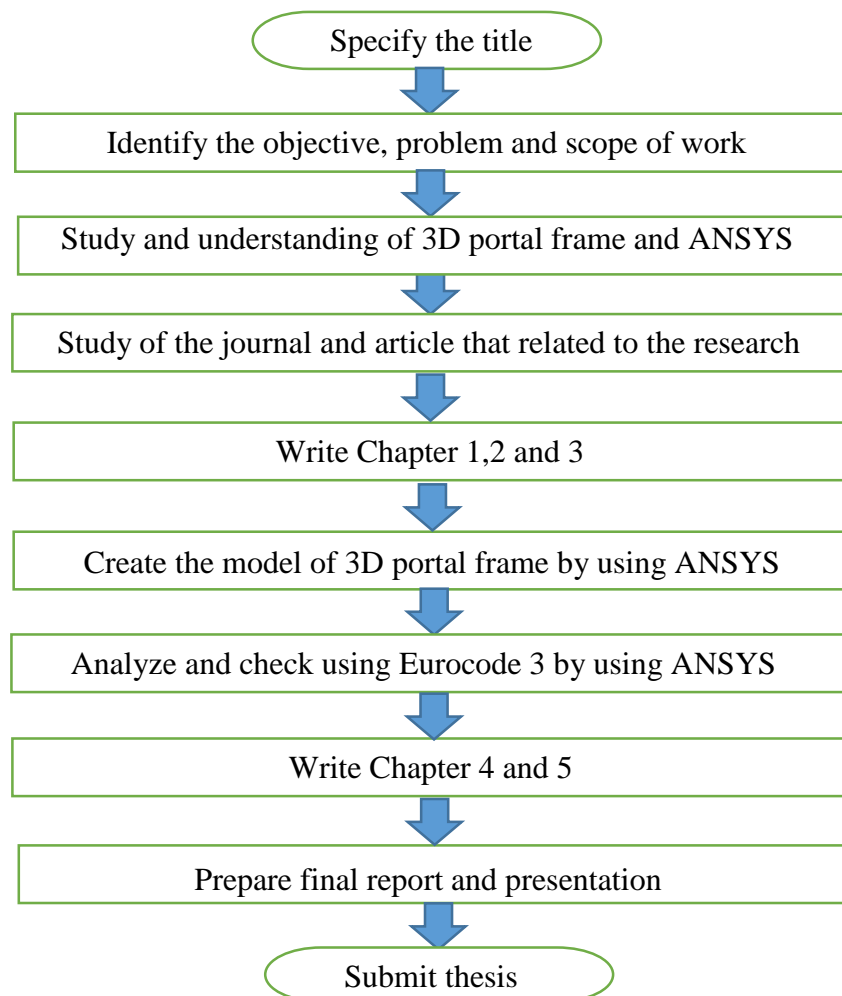
Figure 3.1: Flow Chart of the research process

3.2 PREPROCESSING

In this level, the input data of 3D portal frame was defined accordingly to the Eurocode 3 (EN 1993-1-1:2005). A 3D model of portal frame was created in this level by the following step by step methods.

3.2.1 Entering Title

First of all, the title is needed to be entered for the analysis. To specify a title, it can be done by going to the Utility Menu Bar at the top of the window and select File. Click on the Change Title. Type in the name of the title which is "Portal Frame" and click on "OK". This is shown in Figure 3.2 below. After that, the title will appear at the bottom left corner. CIVILFEM has to be activated in order to specify and check our model according to Eurocode 3. From the Main Menu, select CIVILFEM and click on "OK" to activate it as shown in Figure 3.3 below.
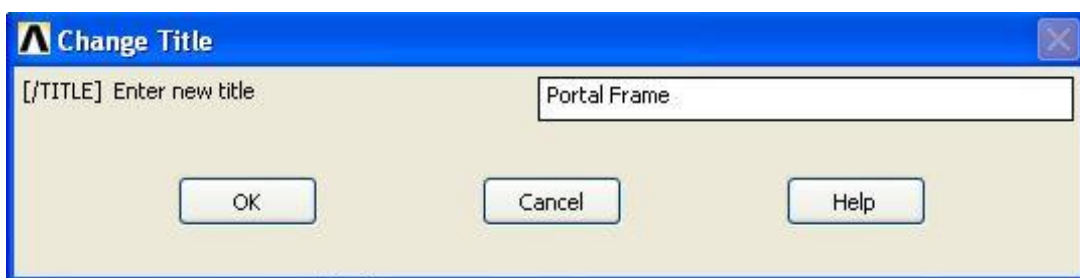
Figure 3.2: Change Title
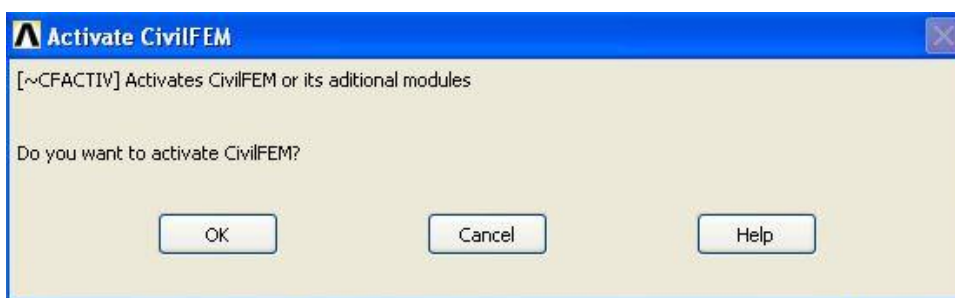
Figure 3.3: Activate CIVILFEM

3.2.2 Set Codes and Units

In this project, the active code that going to be used is Eurocode 3 as the structure that is going to be examined is portal frame which is made up of steel structure although it is allowed to uphold different active codes simultaneously in CIVILFEM. To set the code that going to be used, we have to go to the Main Menu of ANSYS and select Civil Setup. In the window as shown in Figure 3.4 as below, Eurocode 3 (EN 1993-1-1:2005) was selected for the code checking of the steel structure. While for the Unit System going to be used in this project is to be set in the Units column in the same column. International System Units was the choice of system units in this project. However, for Monetary Unit, it is depends on the currency to be used. In this project, Euro was chosen. Click "OK". The CIVILFEM setup option for units is shown in Figure 3.5 below.
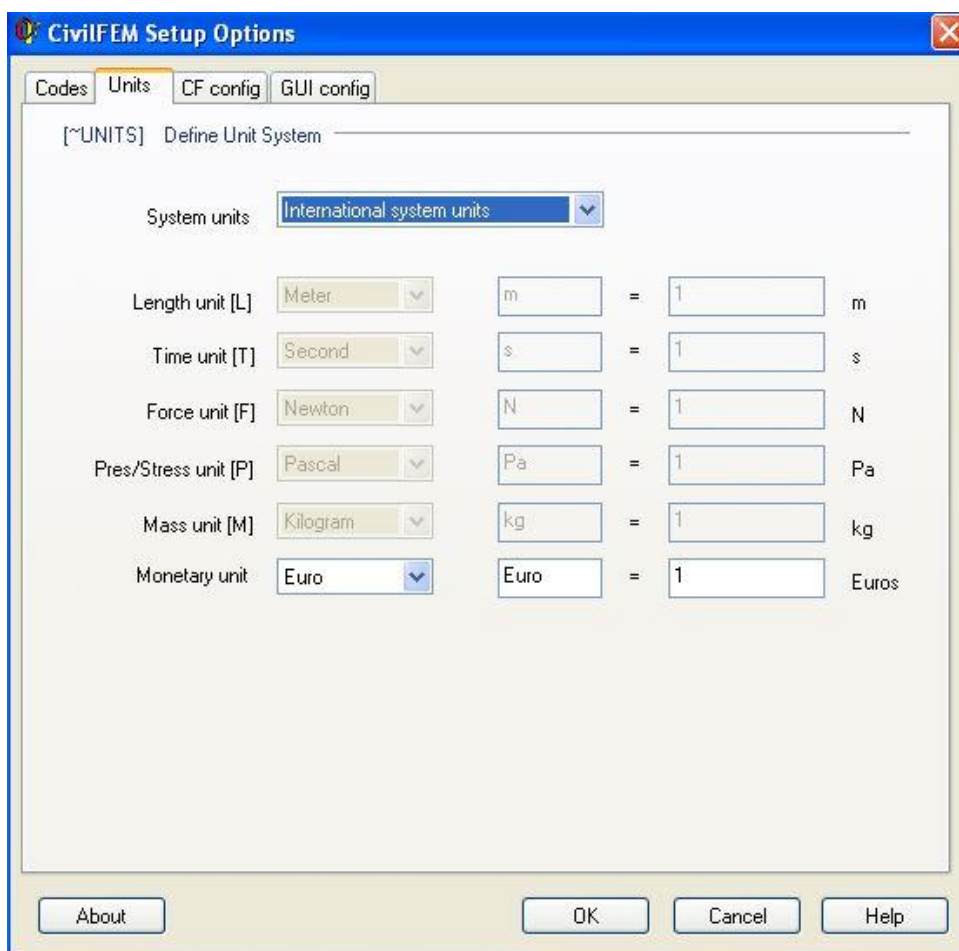
Figure 3.4: CIVILFEM setup options for codes
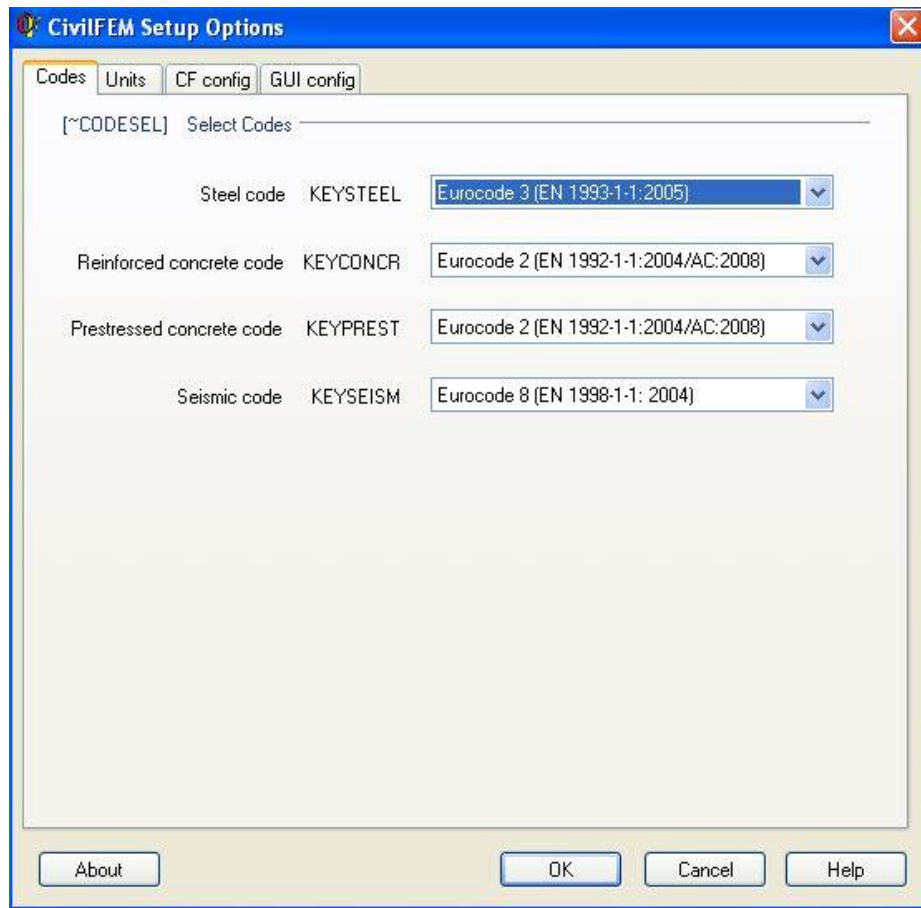
Figure 3.5: CIVILFEM setup options for units

3.2.3 Defining Element Types

With the integration of CIVILFEM in ANSYS, element type of the structure is able to be defined in this computer program in order to let the design and checking by Eurocode 3. From the Main Menu, Civil Preprocessor should be selected followed by Element Type and then Civil Beam. In the window as shown in Figure 3.6 below, we will select 3D Lin. Beam 188.

Lastly, click "OK" to determine the element type. Although there might be different type of element may be used for different location of the element of the strucuture. However, for this structure, 3D Lin. Beam 188 is going to be used for the every element of this structure.
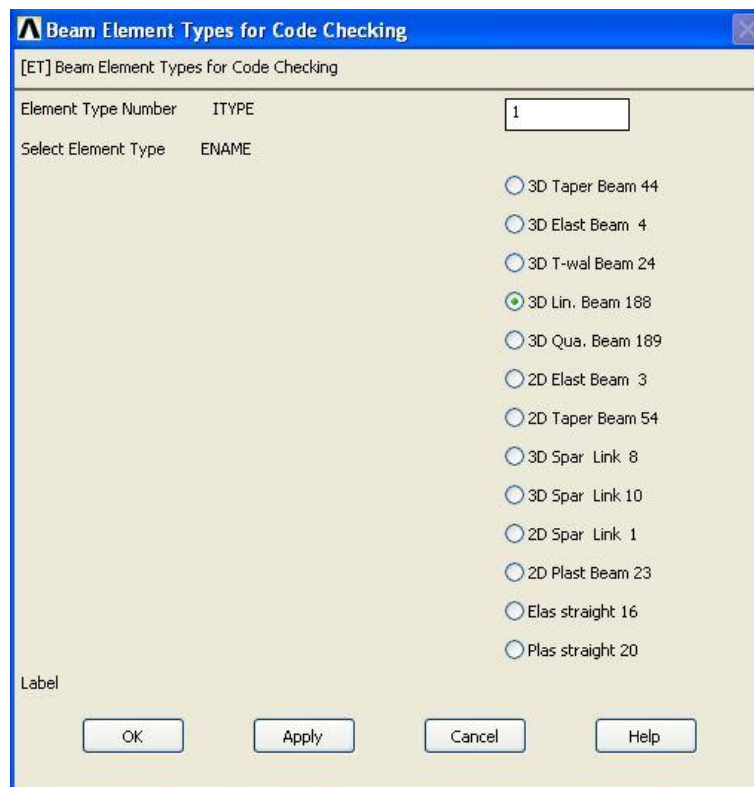
Figure 3.6: Selecting element type

3.2.4 Defining Material

The CIVILFEM ~CFMP command allows user to define stress-strain diagram, safety coefficient and control of linear or non-linear behavior of materials. From the Main Menu, choose Civil Preprocessor. Under Civil Preprocessor, choose Materials. A window will show up named Material Browser and then click on new. After that, another window will show up like shown in the Figure 3.7 as shown below.

There are several types of material shown in the window. Since the structure that going to be examined is portal frame which made up of steel structure, choose Structural Steel. We are going to check and design the structure accordingly to Eurocode 3. Therefore, EC3 will be chosen under the Structural Steel. In this project, the material chosen will be Fe E355. Lastly, click Add and the material will be added to be Material Browser. Click "OK" to confirm the choice of material.
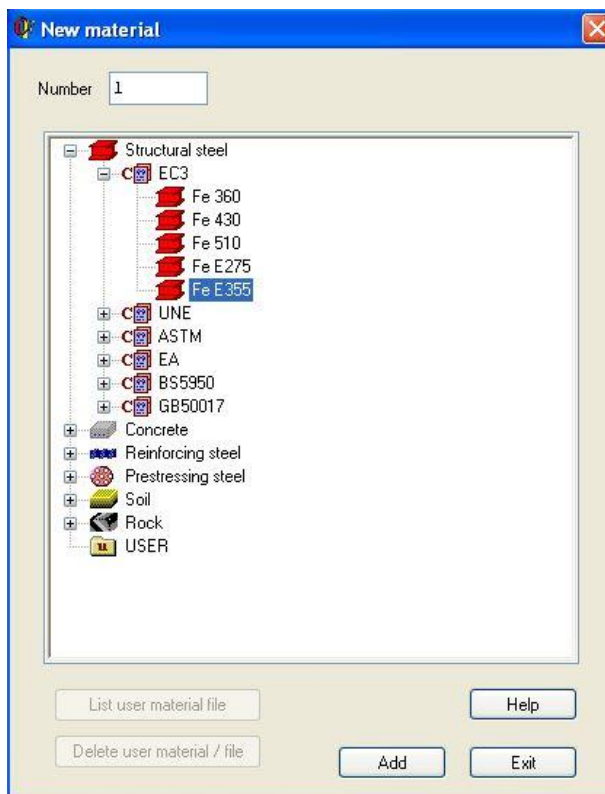
Figure 3.7: New material

3.2.5   Defining Section

From the Main Menu, Civil Preprocessor is selected as usual to create the section of model that is going to be used in this structure. From the column, choose cross section followed by Hot Rolled which can be found at the right hand side of the window. It allows us to enter the Hot Rolled Library of CIVILFEM. In the library, there are a lot of choices of section to be used in the structure but the section of the structure that are going to be examined is under I-Beams. Click on I-Beams followed by IPE A. Choose IPE A 600 as shown in Figure 3.8 below.

After that, click on the tick on the top left corner of the window to confirm the section that going to be used. There is only one section to be used in this project. The member properties is also shown in the window. All element in this structure is assumed to have same type of cross section. Click on Exit to confirm the section.
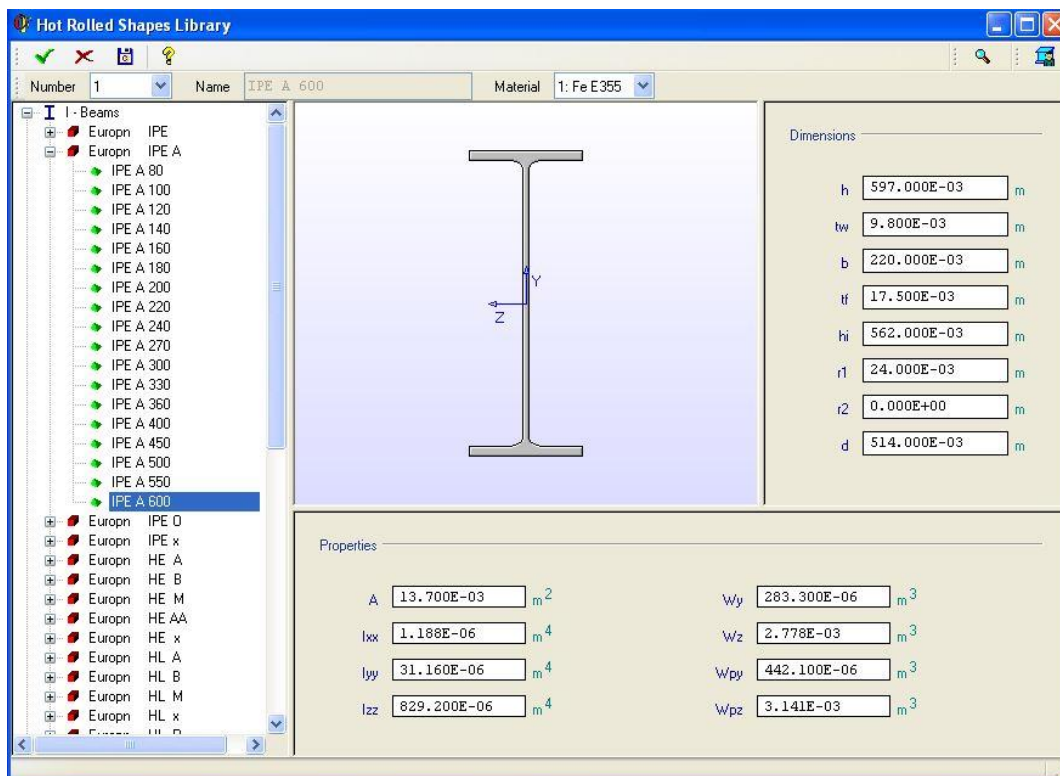
Figure 3.8: Steel cross section

3.2.6 Defining Member Properties

The purpose of defining member property is to check the compression buckling for the compression member. From the Main Menu, as usual, select Civil Preprocessor. Select Member Property under the menu. Click on New Property under the window of Member Property Explorer. Firstly, enter the name of member that going to be keyed in which is "Horizontal Member 1". Enter the length of the member as well which is "3.0m". After that, unmark the "LatBuck" which means not allowed and also choose –Z in "ChckAxis". The same procedure is used to enter the following the other member property with different name and length. These procedure must be repeated until the the last type of element that going to be used in this research has been keyed in. The total number of member property that will be used in this analysis is 17 which is shown in Table 3.2 below. The number of member are allowed to be keyed in as any name as it would not affect the results of the research. The window of Member Property is also shown in Figure3.9 below.

Table 3.2: Member Properties

No.	Name	Length (m)
1	Horizontal Member 1	3.00
2	Vertical Member 1	2.75
3	Diagonal Member 1	1.52
4	Vertical Member 2	11.50
5	Vertical Member 3	12.00
6	Horizontal Member 2	40.00
7	Horizontal Member 3	5.00
8	Horizontal Member 4	10.00
9	Diagonal Member 2	7.87
10	Diagonal Member 5	20.19
11	Diagonal Member 3	11.40
12	Diagonal Member 4	12.37
13	Horizontal Member 5	12.00
14	Vertical Member 4	11.00
15	Diagonal Member 6	12.08
16	Horizontal Member 6	20.00
17	Diagonal Member 7	6.08

Figure 3.9: Member properties 1

3.2.7 Defining Beam & Shell Properties

In this step, beam and shell properties (BMSHPRO) which is one of the CIVILFEM command is used to define ANSYS real constants. As usual, Civil Preprocessor is to be selected from the Main Menu followed by Beam & Shell Properties. Choose New Beam under the window of Beam & Shell Properties. In the window of Beam, firstly, the name of the beam is to be keyed in as Horizontal Member 1. After that, select Beam 188 as the Ename. While for the member properties number, we should select "1" accordingly to the member property which already keyed in in the previous step. Choose "1" on the top of the cross section diagram in the window. ANSYS Real Constant can be shown by clicking "Real Constant". The same procedure are going to be repeated for the following Beam & Shell Properties for 17 times by keying in the different name of member and selecting member properties number respectively. Click the tick after all the properties are done keying in. Lastly, click Exit to confirm. The Beam & Shell Properties to be used in this analysis is shown in Table 3.3 below while the window of beam is shown in Figure 3.10 below as well.
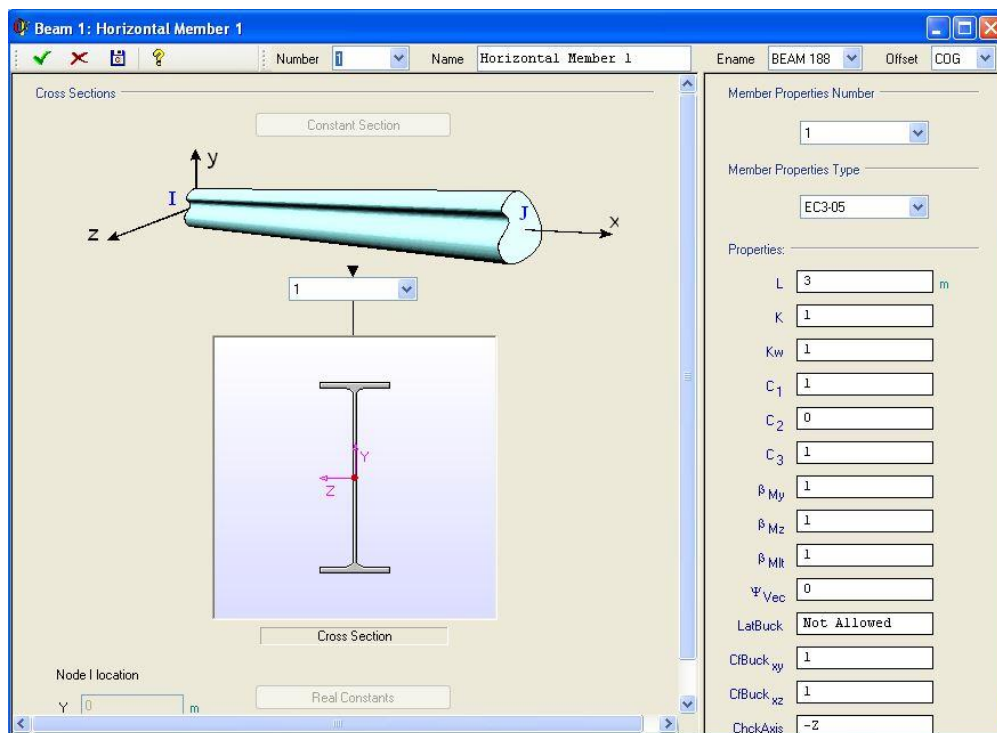
Figure 3.10: Beam 1: Horizontal Member 1

Table 3.3: Beam & Shell Properties

No.	Name	Member Properties Number
1	Horizontal Member 1	1
2	Vertical Member 1	2
3	Diagonal Member 1	3
4	Vertical Member 2	4
5	Vertical Member 3	5
6	Horizontal Member 2	6
7	Horizontal Member 3	7
8	Horizontal Member 4	8
9	Diagonal Member 2	9
10	Diagonal Member 5	10
11	Diagonal Member 3	11
12	Diagonal Member 4	12
13	Horizontal Member 5	13
14	Vertical Member 4	14
15	Diagonal Member 6	15
16	Horizontal Member 6	16
17	Diagonal Member 7	17

3.2.8   Defining Nodes and Elements

In order to define the nodes and elements of the structure, Preprocessor is to be selected in the Main Menu. After that, Modeling, Create and lastly Node will be selected accordingly. Pick Active CS and then enter "1" as node 1 and enter 0,0,0 as the first coordinate of the structure. Figure 3.11 shows the window of Create Nodes in Active Coordinate System. Pick Apply and continue to key in the coordinate for the rest of the nodes. The same method is used to key in the rest of the nodes. There are total of 81 nodes to be keyed in for this structure. Table 3.4 and Table 3.5 are showing the nodes with their coordinates respectively. Figure 3.12 is showing the total of 81 nodes plotted. The elements are to be joined in the next step.
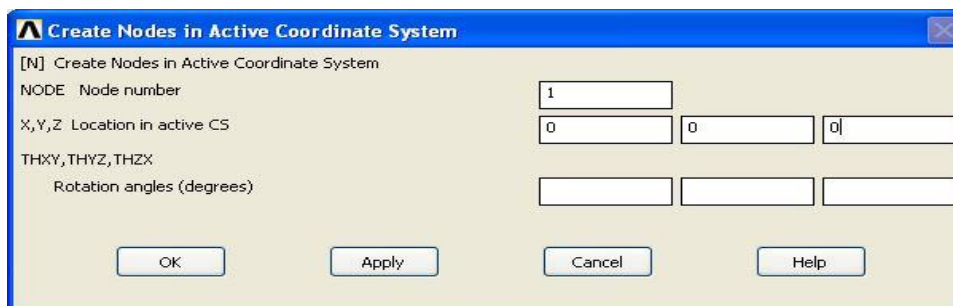
Figure 3.11: Create Nodes in Active Coordinate System

Table 3.4: Nodes with Coordinates

Node	Coordinate	Node	Coordinate
1	0,0,0	21	0,11,40
2	0,0,10	22	6,12,40
3	0,0,20	23	12,11,40
4	0,0,30	24	0,0,40
5	12,0,0	25	12,0,40
6	12,0,10	26	7.5,11.75,40
7	12,0,20	27	7.5,11.75,0
8	12,0,30	28	9,11.5,0
9	0,11,0	29	9,11.5,40
10	0,11,10	30	10.5,11.25,40
11	0,11,20	31	10.5,11.25,0
12	0,11,30	32	1.5,11.25,0
13	6,12,0	33	1.5,11.25,40
14	6,12,10	34	3,11.5,0
15	6,12,20	35	3,11.5,40
16	6,12,30	36	4.5,11.75,0
17	12,11,0	37	4.5,11.75,40
18	12,11,10	38	12,2.75,40
19	12,11,20	39	12,2.75,0
20	12,11,30	40	12,5.5,40

Table 3.5: Nodes with Coordinates

Node	Coordinate	Node	Coordinate
41	12,5.5,0	61	6,0,40
42	12,8.25,0	62	0,0,5
43	12,8.25,40	63	0,0,15
44	0,2.75,0	64	0,0,25
45	0,2.75,10	65	0,0,35
46	0,5.5,10	66	0,11,5
47	0,5.5,0	67	0,11,15
48	0,8.25,0	68	0,11,25
49	0,8.25,10	69	0,11,35
50	0,2.75,30	70	6,12,5
51	0,5.5,40	71	6,12,15
52	0,5.5,30	72	6,12,25
53	0,8.25,30	73	6,12,35
54	0,8.25,40	74	12,11,5
55	0,2.75,40	75	12,11,15
56	3,0,0	76	12,11,25
57	9,0,0	77	12,11,35
58	3,0,40	78	12,0,5
59	9,0,40	79	12,0,15
60	6,0,0	80	12,0,25
		81	12,0,35
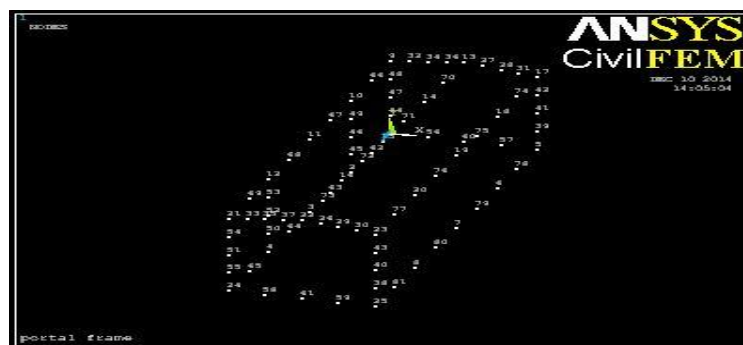
Figure 3.12: Total of 81 nodes

3.2.9 Creating of Model

The elements in the structure have their own length. Therefore we are using element attributes to classify every length of element. Select Preprocessor from the Main Menu. After that, select Modeling, Create, Element and lastly Elem Attributes. Element type number is always "1" as only Beam188 is used as the element in this project. Section number need to be selected according to the element needed to draw. "1 RolledI" is selected as the section number. This section means that the element length is 3m and named Horizontal Member 1. Figure 3.13 shows the element attributes window.

After the element attribute is set, element can be created by selecting Preprocessor from the Main Menu, Modeling, Create, Element, Auto Numbered and then Thru Nodes. All the horizontal members with 3m length are connected by node to node. The same steps are repeated until 17 element attributes are used and lastly the model is created. Figure 3.14 shows the model of 3D portal frame.
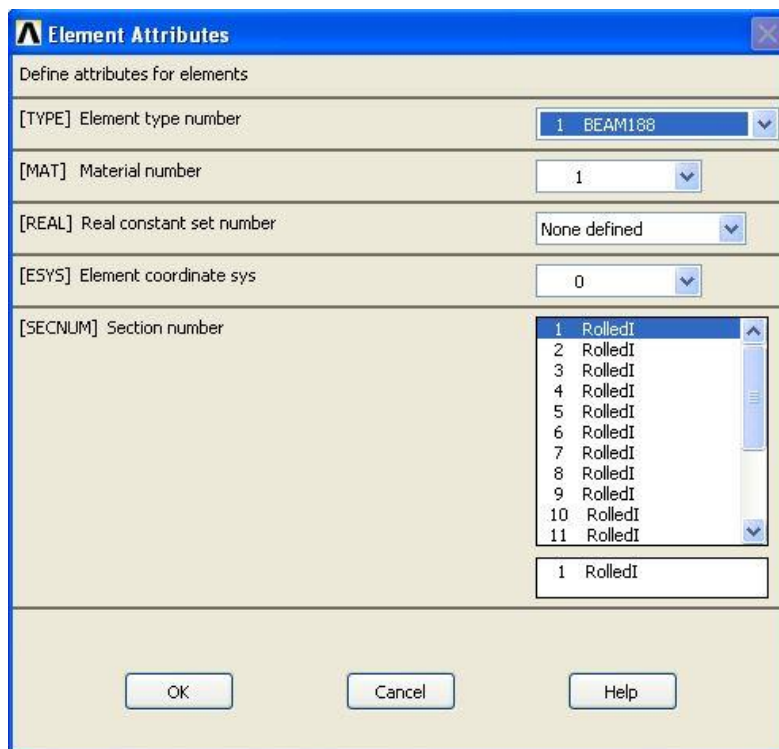
Figure 3.13: Element Attributes
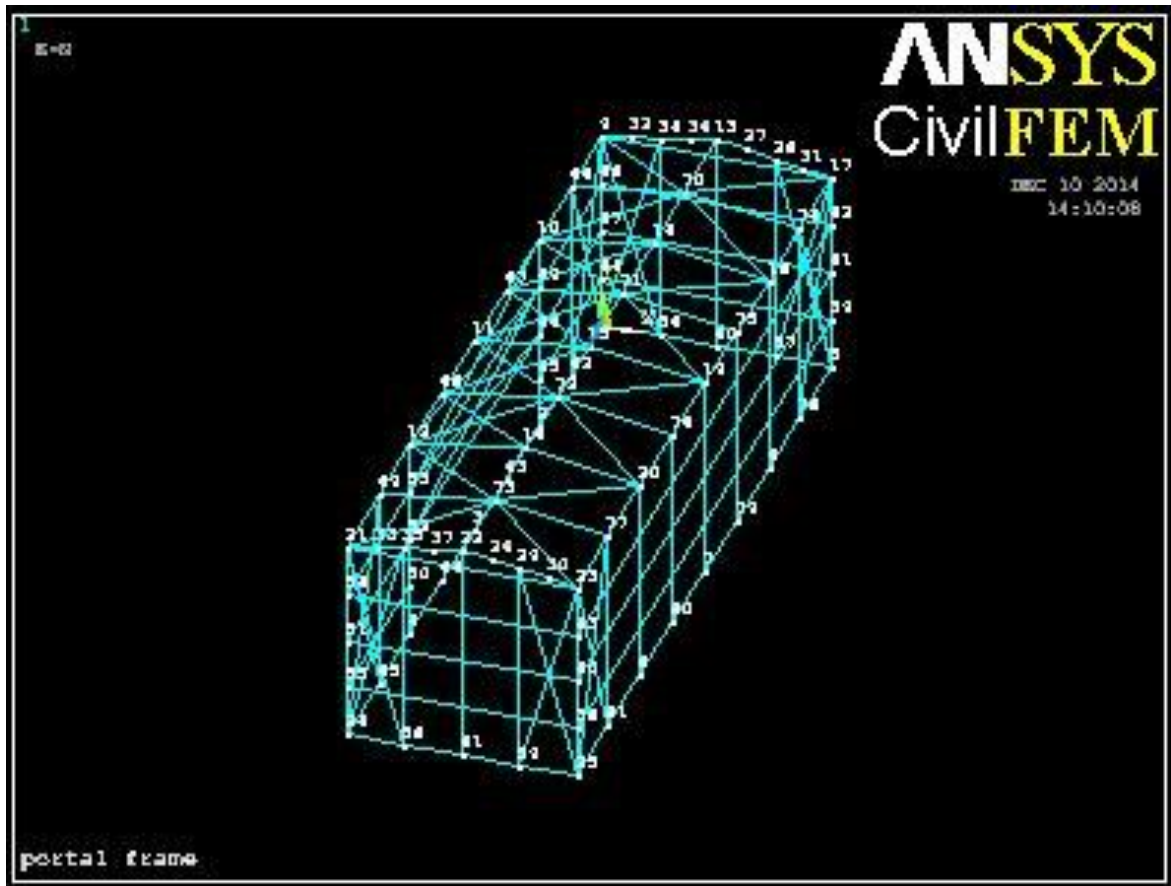
Figure 3.14: Model of 3D portal frame

3.3   SOLUTION PHASE

This phase is very important to analyze the structure before generating the results at the post processing phase. This phase is where the structure will be assigned forces and setting of constraint. After that, solve the analysis.

3.3.1   Define Analysis Type

At the Main Menu, select Analysis Type and click on New Analysis as shown in Figure 3.15 below. Select Static as the analysis type of the structure that going to be analyzed. Lastly, click "ok" to confirm the selection.
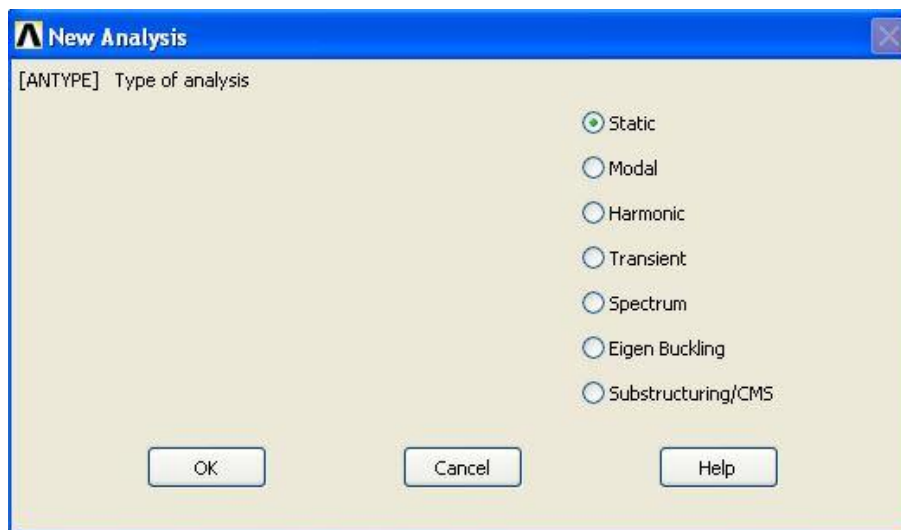
Figure 3.15: Type of Analysis

3.3.2 Apply Constraints

From Main Menu, select Solution. Under Solution, choose Define Loads, Apply, Structural, Displacement and lastly choose On Node. Pick node 1, 2, 3, 4, 5, 6, 7, 8, 24, 25, 62, 65, 78 and 81. Choose All DOF for the DOFs to be constrained and enter "0" for the displacement value. Figure 3.16 shows the window of Apply U,ROT on Nodes. Other than that, Figure 3.17 is showing the model that has been applied displacement constraint on it.
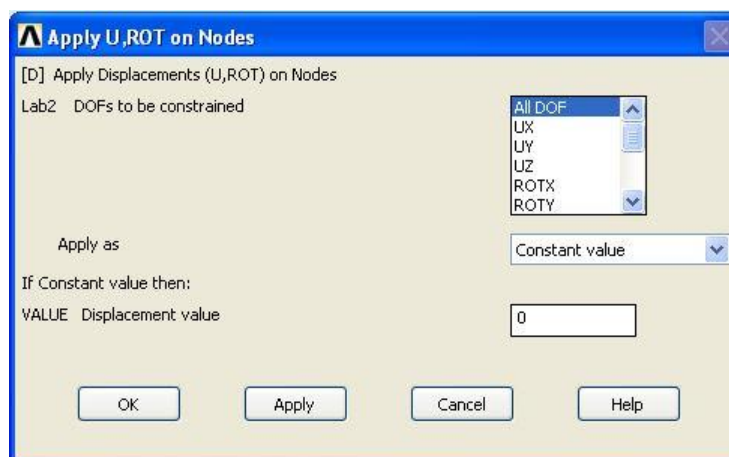
Figure 3.16: Apply U,ROT on Nodes
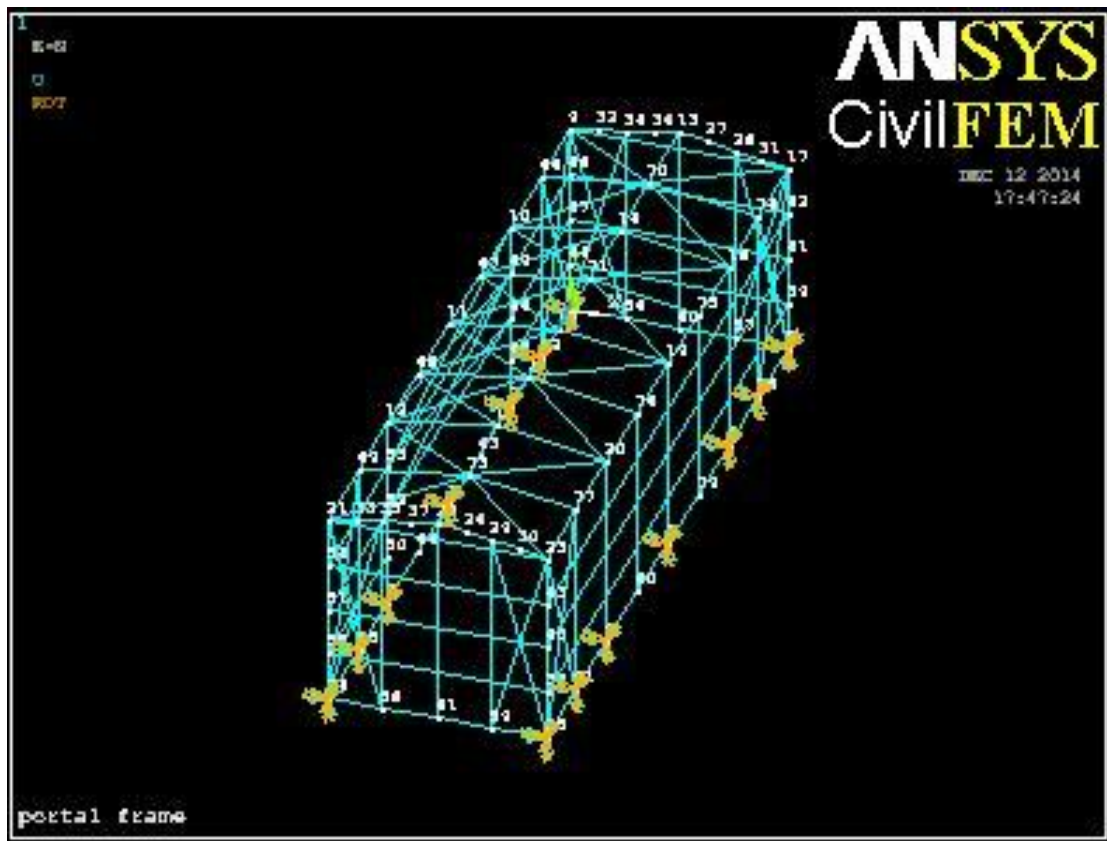
Figure 3.17: Displacement constraint applied on model

3.3.3   Apply Loads

From Solution Menu, choose Define loads, Apply, Structural, Force/Moment lastly choose On Nodes. Pick on the node 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19, 20, 21, 22, 23, 66, 69, 70, 73, 74 and 77. In order to apply the load vertically, choose FZ for the direction of force/mom and key in "-50000" for the force/moment value. The negative sign is meaning the direction of the loads which means the loads are pointing the opposite side of the y-direction. They are pointing downward. There are total 21 of point loads acting on the structure. Figure 3.18 is showing the window of Apply F/M on Nodes. The point loads are including dead loads and live loads which had converted to the form of point load.

The next load which is going to be applied is wind load. Assume the load is acting on one side only. From Solution Menu, choose Apply, Structural, Pressure, On Beams.

Choose the elements that are going to sustain the wind load. After that, in the window of Apply PRES on Beams as shown in Figure 3.19 below, enter "0.8*15000" as the wind load at the Pressure value at node I. In the mean while, Figure 3.20 is showing the model that has been applied both point loads and wind loads.
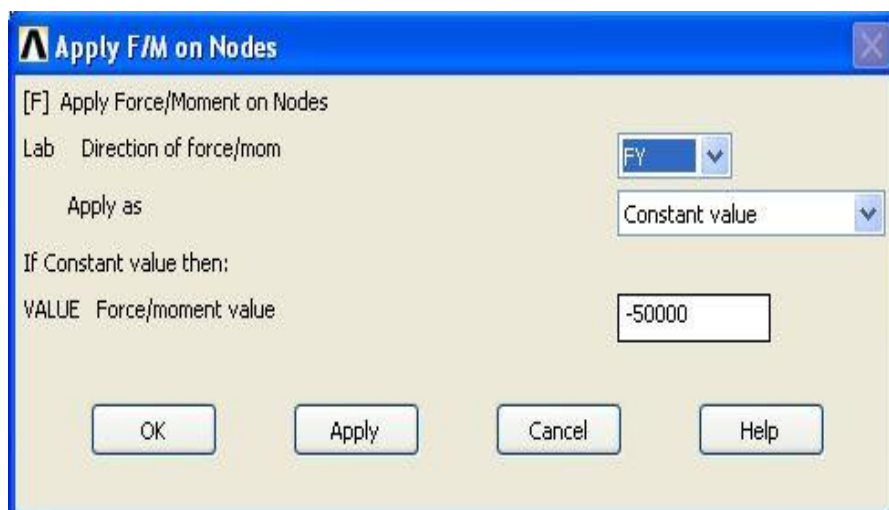
Figure 3.18: Apply F/M on nodes
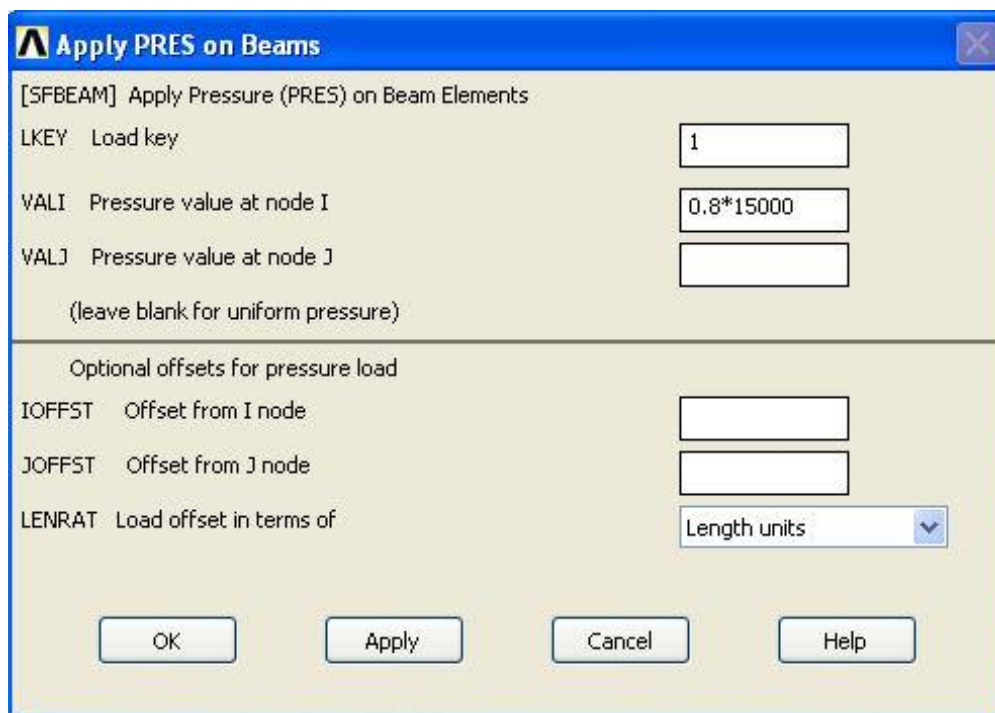
Figure 3.19: Apply PRES on Beams
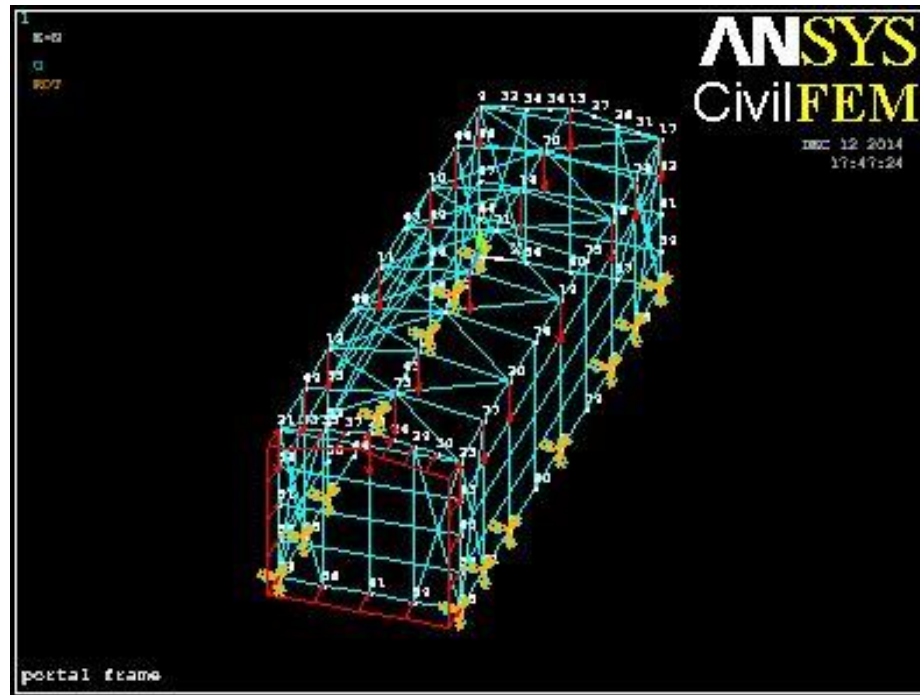
Figure 3.20: Model that has been applied point loads and wind load

3.3.4 Solving

In the Solution Menu, choose Solve then select Current LS to begin the solution of current Load Step (LS). Click OK to start the solution as shown in Figure 3.21 and the when the solution is done, the window will pop out to show that solution is done as shown in Figure 3.22.

Figure 3.21: Solve Current Load Step
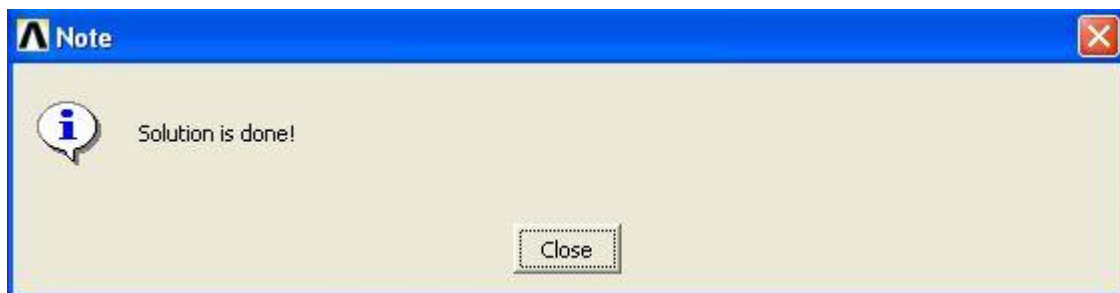
Figure 3.22: Note

3.4   POSTPROCESSING

In postprocessing, general results of reaction forces, deformed shape, deflection, axial forces, shear stress, bending moment and code checking by using Eurocode 3.

3.4.1   Reaction Forces

From the Main Menu, General Postproc, choose plot results and Deformed Shape. Pick "Def + Under Edge" for the deformed shape to be plotted and lastly click on "OK". Figure 3.23 shows the window of Plot Deformed Shape.
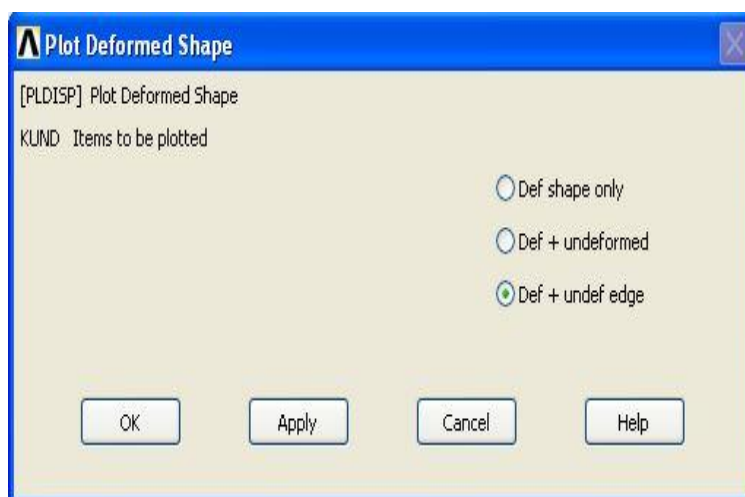
Figure 3.23: Plot Deformed Shape
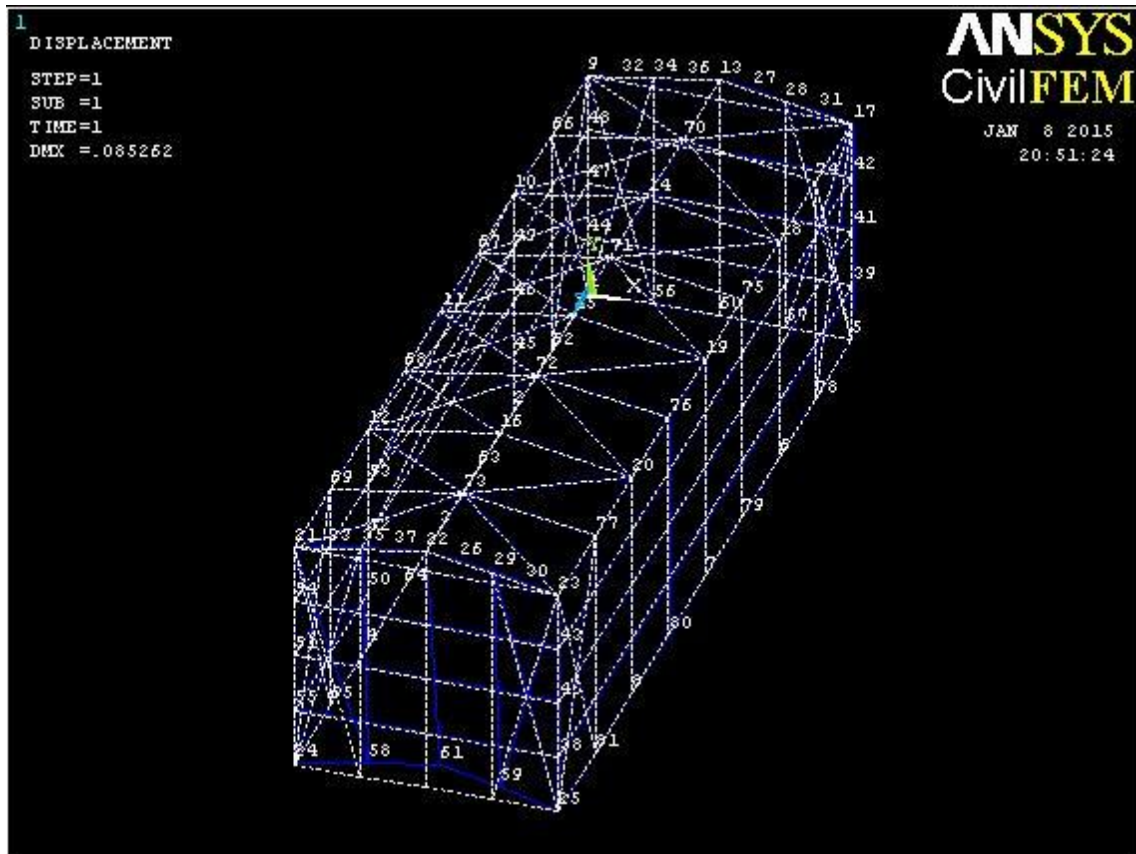
Figure 3.24: Deformed Shape

Figure 3.24 shows the overall deformation of the structure. Deformation is the shape of a structure will become after forces are applied to them. Elastic deformation is the ability of a structure to return back to its original shape after the forces are removed while plastic deformation is would not return back to its original shape. The deformation of the structure shows that the structure is tend to move to negative of z-direction as the force are stronger to that side.

3.4.2 Deflection

As usual, General Postproc is chosen at the Main Menu then followed by Plot Results and Contour Plot. The window as shown in Figure 3.24 will come out and choose Nodal solution then lastly Displacement Vector Sum. Click "OK" to plot the results. Displacement is an important checking which the simulation of the real condition of the

structure will deflect can be seen through this process. The deflection will be plotted in contour which we can define the critical part of the structure by colour of it.
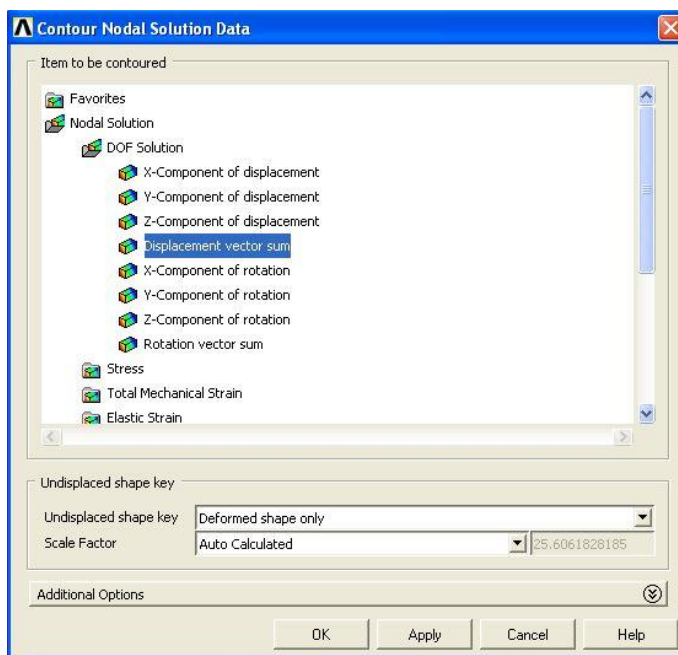
Figure 3.25: Contour Nodal Solution Data
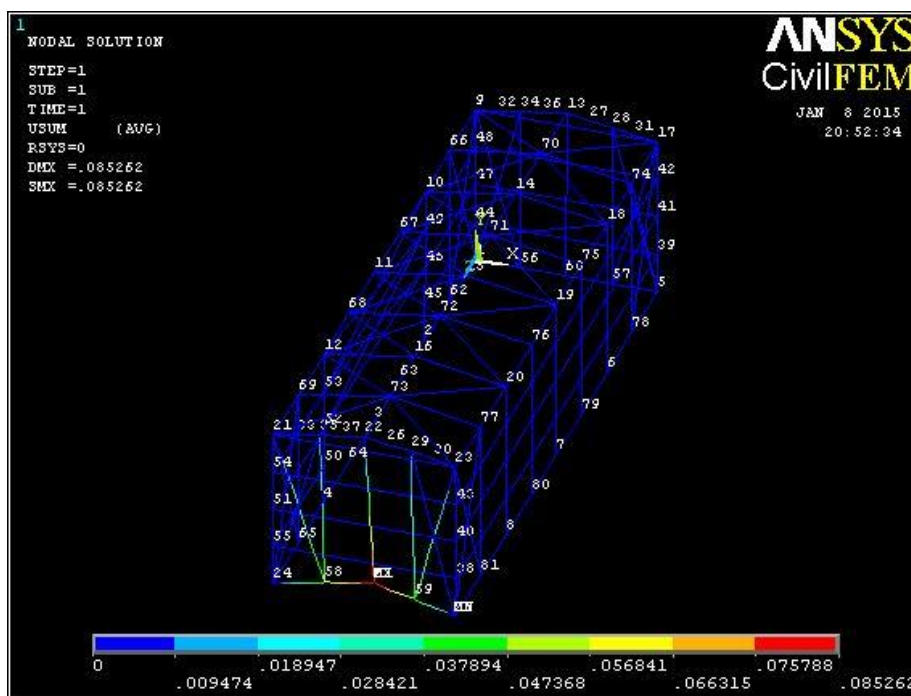
Figure 3.26: Contour Plot of Deflection

Figure 3.26 shows the displacement vector sum for each member. It combined the displacement in every direction and formed vector sum. The highest displacement in the figure is 85.262mm. The blue color structure is the part that having the lesser displacement. The contour plot shows the range of displacement from 0mm until 85.262mm. From the figure, mostly of the parts are having very small displacement only.

3.4.3   Read Results

This step is required to read results for the following checking by CIVILFEM. This is can be accessed from Civil Postprocessor, Read Results, By Load Step. Figure 3.25 shows the window pop out and click on "OK".
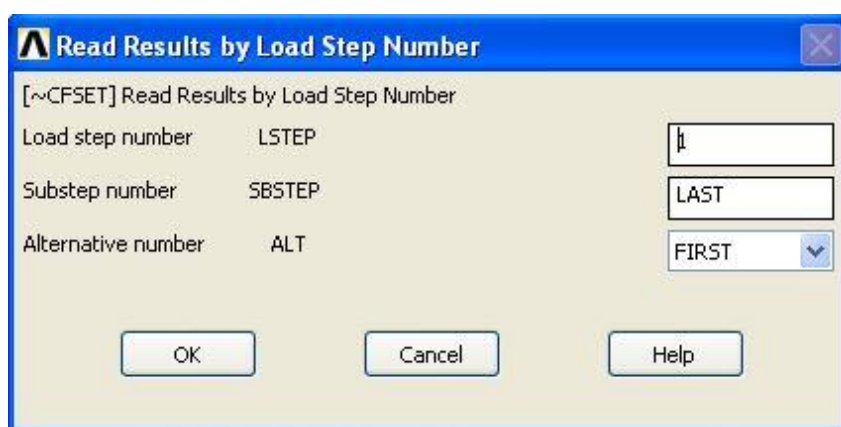
Figure 3.27: Read Results by Load Step Number

3.4.4   Forces & Moments

In this step, axial force, shear force and bending moment are able to be plotted. From General Postproc, choose Beam Utilities, Graph Results and lastly Forces & Moments. As shown in Figure 3.26, Axial Force X is chosen to plot the results for Axial Force. These steps are going to be repeated to plot Shear and Bending Moments.
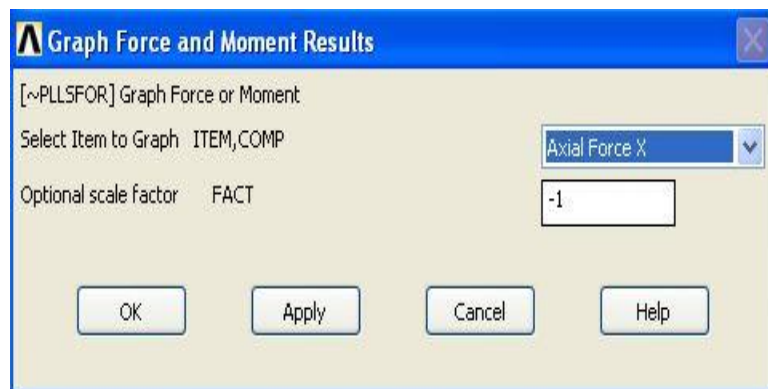
Figure 3.28: Graph Force and Moment Results
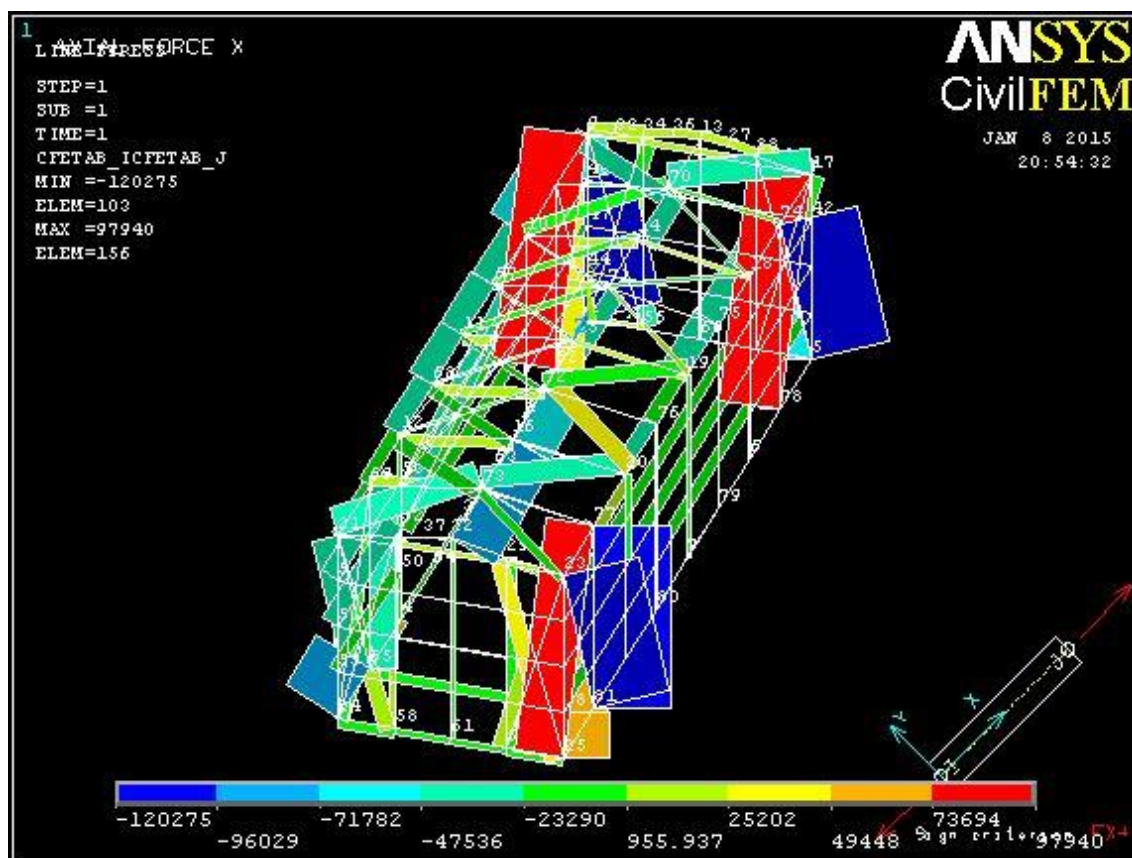
Figure 3.29: Axial Force Diagram

Figure 3.29 shows the axial forces in each member of the whole structure. The axial forces in the member are ranged from -120275N to 73694N. The contour indicates different axial forces between the ranges.
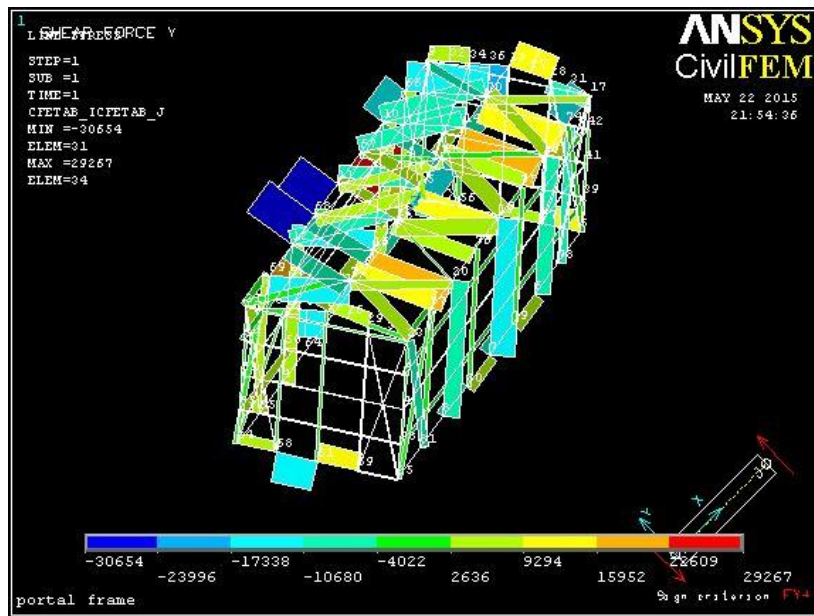
Figure 3.30: Shear Force Diagram

Figure 3.30 shows the shear forces in each member of the whole structure. The shear forces in the member are ranged from -30664N to 22609N. The contour indicates different shear forces between the ranges. However, the highest shear force in the structure is until 10402N only.
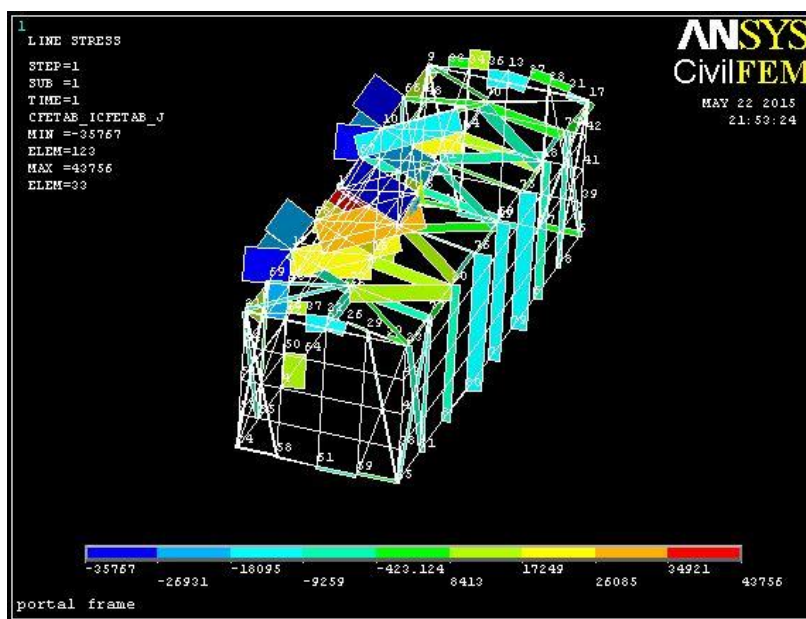
Figure 3.31: Bending Moment Diagram

Figure 3.31 shows the bending moments in each member of the whole structure. The bending moments in the member are ranged from -35767N to 43756N. The contour indicates different bending moments between the ranges.

3.4.5   Code Checking

Code checking is the most important checking for a steel structure. In this structure, only Eurocode 3 will be used to check it. It can be accessed from the Main Menu, click on Civil Postprocessor, Code Checking, Eurocode 3, Check by Code and lastly Tension. Figure 3.27 shows the window to check model results by Eurocode 3. Choose CIVILFEM –Z Axis and click "OK". This allow CIVILFEM to analyze the structure according to Eurocode 3. After that, under Beam Results also, the results are ready to be plotted, click on Plot Results and the window pop out is as shown in Figure 3.28. Choose Tension and Element OK. Click "OK' to confirm the results that need to be plotted. Repeat these steps for compression, bending, shear, compression buckling and lateral buckling.

Figure 3.32: Check Model Results By Eurocode 3
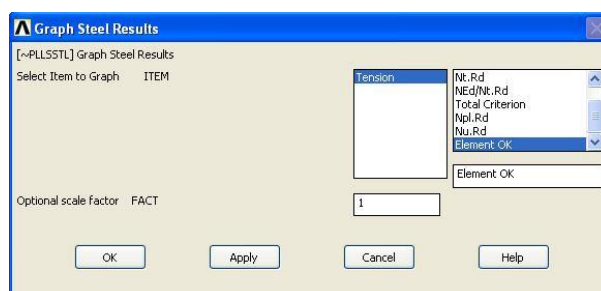
Figure 3.33: Graph Steel Results
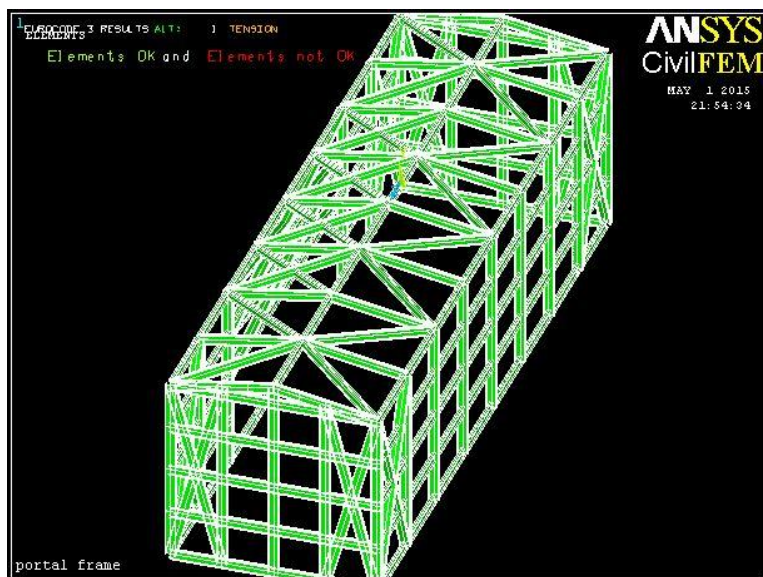
Figure 3.34: Tension Checking Results

Figure 3.34 above shows the results of checking of tension by using Eurocode 3. The checking results are shown in one figure only because the results of all checking are the same in this research. The figure shows that the whole structure is passing all the checking as green colour symbolized element is ok while red colour symbolized the element is fail. All element is ok for all the checking.
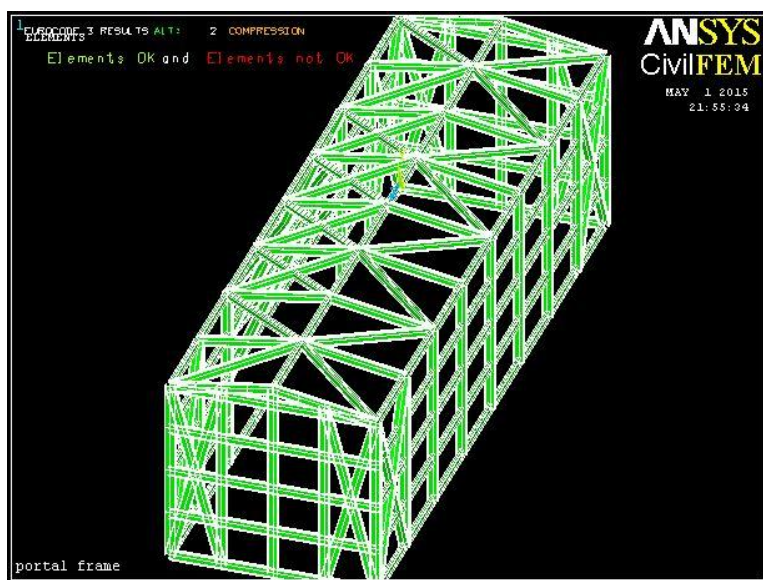
Figure 3.35: Compression Checking Results

Figure 3.35 above shows the results of checking of compression by using Eurocode 3. The checking results are shown in one figure only because the results of all checking are the same in this research. The figure shows that the whole structure is passing all the checking as green colour symbolized element is ok while red colour symbolized the element is fail. All element is ok for all the checking.
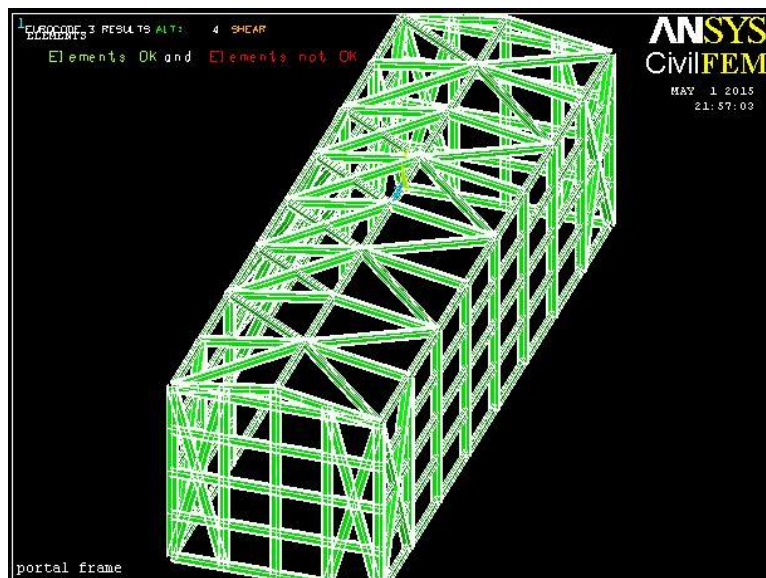
Figure 3.36: Shear Resistance Checking Results

Figure 3.36 above shows the results of checking of shear resistance by using Eurocode 3. The checking results are shown in one figure only because the results of all checking are the same in this research. The figure shows that the whole structure is passing all the checking as green colour symbolized element is ok while red colour symbolized the element is fail. All element is ok for all the checking.

Figure 3.37 below shows the results of checking of bending moment resistance by using Eurocode 3. The checking results are shown in one figure only because the results of all checking are the same in this research. The figure shows that the whole structure is passing all the checking as green colour symbolized element is ok while red colour symbolized the element is fail. All element is ok for all the checking.
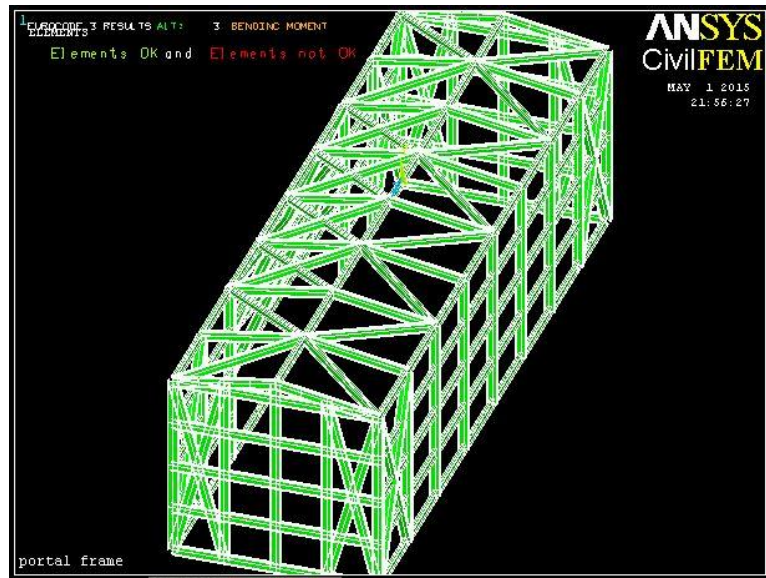
Figure 3.37: Bending Moment Resistance Checking Results
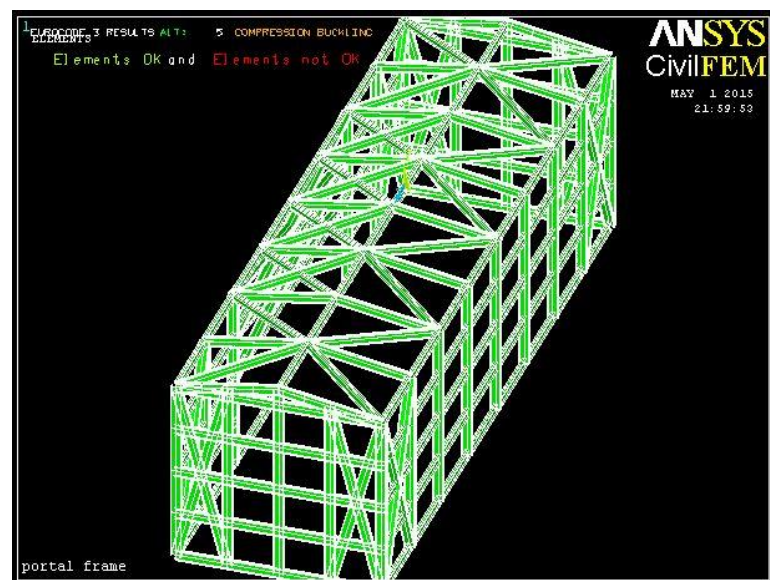
Figure 3.38: Compression Buckling Checking Results

Figure 3.38 above shows the results of checking of compression buckling by using Eurocode 3. The checking results are shown in one figure only because the results of all checking are the same in this research. The figure shows that the whole structure is passing all the checking as green colour symbolized element is ok while red colour symbolized the element is fail. All element is ok for all the checking.
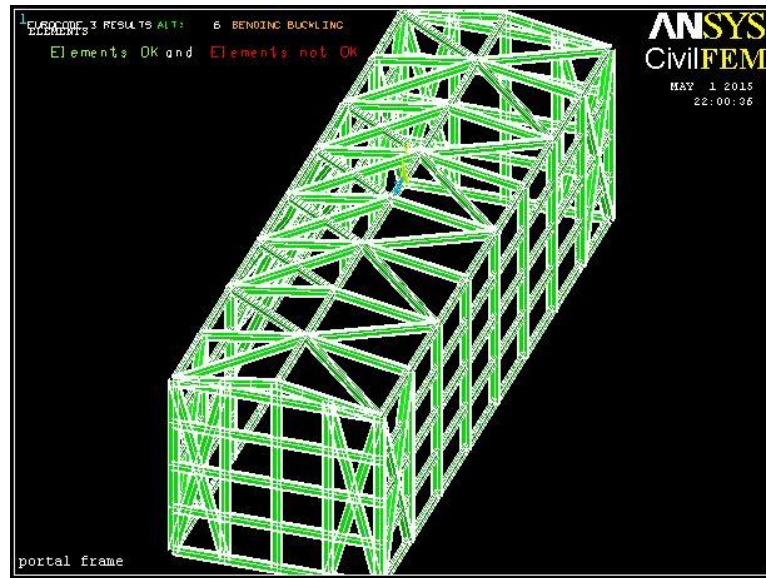
Figure 3.39: Lateral Torsional Buckling Checking Results

Figure 3.39 above shows the results of checking of lateral torsional buckling by using Eurocode 3. The checking results are shown in one figure only because the results of all checking are the same in this research. The figure shows that the whole structure is passing all the checking as green colour symbolized element is ok while red colour symbolized the element is fail. All element is ok for all the checking.

CHAPTER 4

RESULTS AND DATA ANALYSIS

4.1 INTRODUCTION

In this chapter, we will discuss about the results extracted from the simulation of 3D portal frame model using ANSYS software and will be included with the analysis of graph from the random input and output produced from the simulation. Data analysis is the process of evaluating data using analytical and logical reasoning to examine each component of the data provided. Data from the graph are gathered and then analyzed to form some sort of finding or conclusion. There are two sources of data which are coming from input parameter and output parameter graph. There are two graphs in input parameter which are probability density and probability while in output parameter the graph will show the maximum value of the output parameter. In this 3D portal frame model, there are six input parameters which are, vertical load, wind load, elastic modulus, poison ratio, temperature and density.

Probabilistic analysis result that got in ANSYS was included Statistics of the Probabilistic Results, Sample History Plots, Histogram Plots, Cumulative Distribution Function Plots and Sensitivity Plots. Total 30 number of samples and 10000 numbers of simulations were used in this analysis to get the maximum output of the displacement, axial stress, and axial elastic strain.

4.2 PROBABILISTIC RESULTS FROM ANSYS

4.2.1 Random Input Variables

Table 4.1 shown below are the input variables of this probabilistic analysis to determine the effect of the changing of the variables to the model structure. The symbol of input used in this analysis are called ELASTIC, POISON, LOAD, WINDLOAD, TEMP and DENS. As you can see from the table, variable DENS is Gaussian variable with Par1 that indicating mean value of 7850 kg/m^3 and Par2 3000 kg/m^3 that used in this research. Same for other variables.

TABLE 4.1: Random Input Variables Specifications

No.	Name	Type	Par1	Par2
1	DENS	GAUS	7850	3000
2	ELASTIC	GAUS	2.0E+11	1.0E+11
3	POISON	GAUS	0.30	0.25
4	LOAD	GAUS	-25000	500
5	WINDLOAD	GAUS	7500	500
6	TEMP	GAUS	100	80

Probabilistic density function (PDF) and cumulative distribution function (CDF) of input random variable applied load was shown below. The difference between the two graphs is the y-axis will be probability density and probability respectively. According to the Figure 4.1 as shown below, from probability density graph we can see that when the probability design is zero, then the density is 5000 kg/m^3. In the meanwhile, the maximum point of probability density graph is showing 0.0004 and density is around 7600 kg/m^3. The probability graph is calculated by 20, 50 and 80 percent of the density. The percentage value calculated are joined together in order to get the curve for the probability graph as shown.
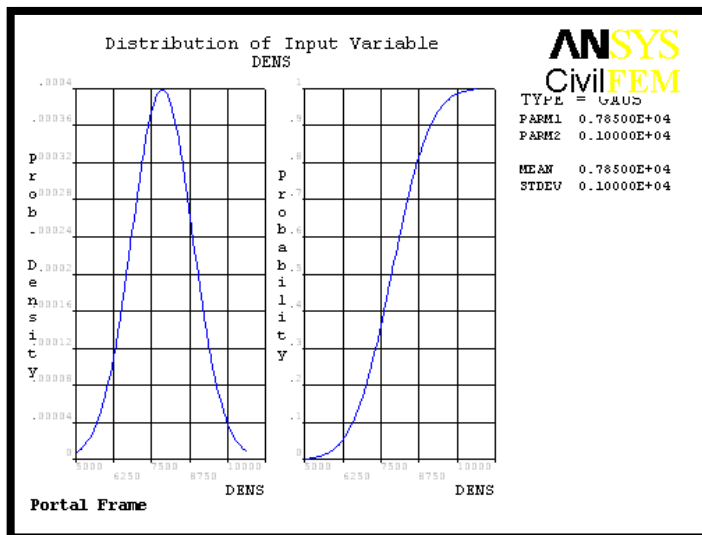
Figure 4.1: PDF & CDF of Input Random Variable DENS

According to the Figure 4.2 as shown below, from probability density graph we can see that when the probability design is zero, then the elastic modules is 70GPa. In the meanwhile, the maximum point of probability density graph is showing 0.00008 and elastic modulus is around 270GPa. The probability graph is calculated by 20, 50 and 80 percent of the density. The percentage value calculated are joined together in order to get the curve for the probability graph as shown.
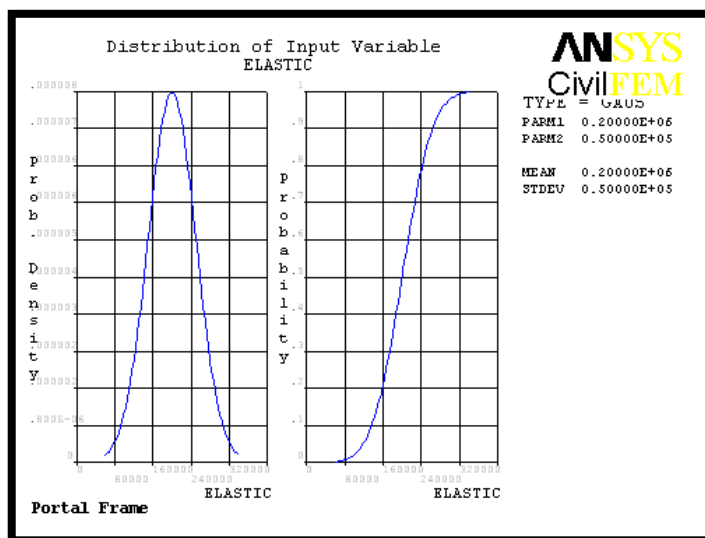
Figure 4.2: PDF & CDF of Input Random Variable ELASTIC

According to the Figure 4.3 as shown below, from probability density graph we can see that when the probability design is zero, then the poison ratio is around -0.4. In the meanwhile, the maximum point of probability density graph is showing 1.6 and poison is around 0.2. The probability graph is calculated by 20, 50 and 80 percent of the density. The percentage value calculated are joined together in order to get the curve for the probability graph as shown.
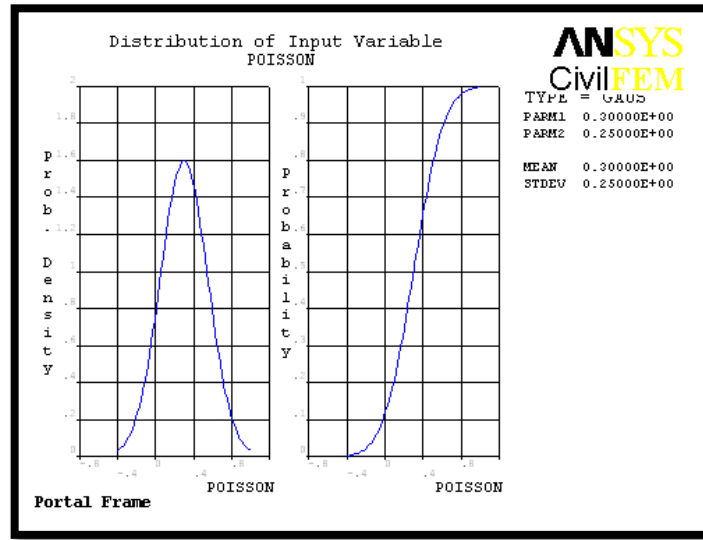
Figure 4.3: PDF & CDF of Input Random Variable POISON
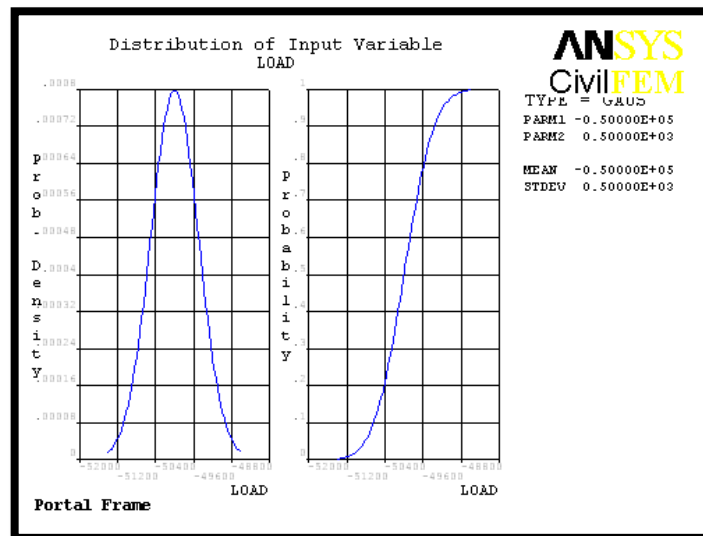
Figure 4.4: PDF & CDF of Input Random Variable LOAD

According to the Figure 4.4 as shown above, from probability density graph we can see that when the probability design is zero, then the vertical load is around -51000kN. In the meanwhile, the maximum point of probability density graph is showing 0.0008 and poison is around -50000kN. The probability graph is calculated by 20, 50 and 80 percent of the density. The percentage value calculated are joined together in order to get the curve for the probability graph as shown.
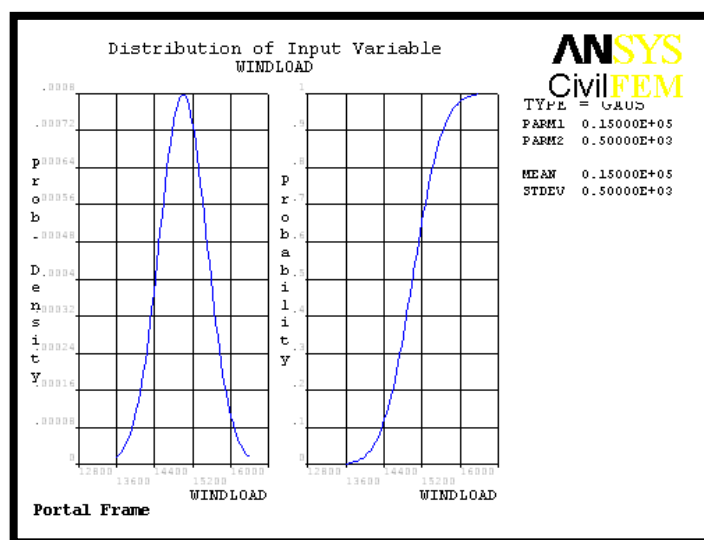
Figure 4.5: PDF & CDF of Input Random Variable WINDLOAD

According to the Figure 4.5 as shown above, from probability density graph we can see that when the probability design is zero, then the wind load is around 13000kN. In the meanwhile, the maximum point of probability density graph is showing 0.0008 and poison is around 15000kN. The probability graph is calculated by 20, 50 and 80 percent of the density. The percentage value calculated are joined together in order to get the curve for the probability graph as shown.
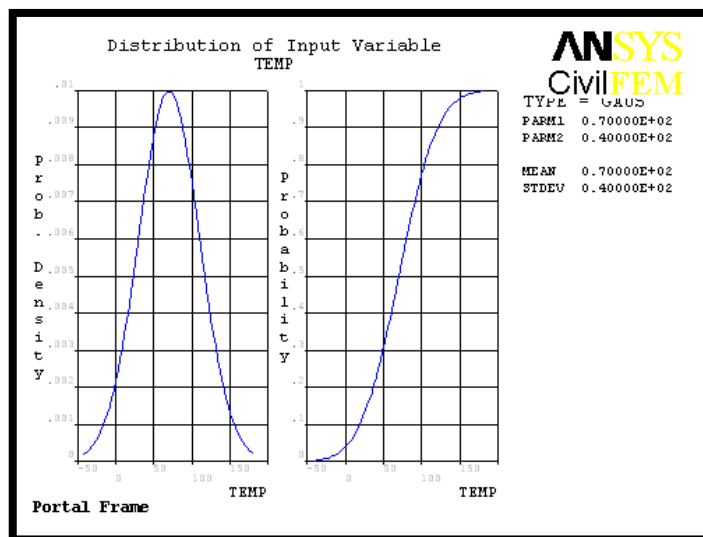
Figure 4.6: PDF & CDF of Input Random Variable TEMP

According to the Figure 4.6 as shown above, from probability density graph we can see that when the probability design is zero, then the temperature is around -50 degree celcius. In the meanwhile, the maximum point of probability density graph is showing 0.01 and poison is around 70 degree celcius. The probability graph is calculated by 20, 50 and 80 percent of the density. The percentage value calculated are joined together in order to get the curve for the probability graph as shown.

4.2.2 Statistics of The Probabilistic Results

ANSYS probabilistic design system analysis has analysis the structure through looping 1000 sample points. The statistical analysis of the input parameter are shown in Table 4.2 while Table 4.3 is showing the output parameter. The tables are showing the mean, standard deviation, skewness, kurtosis, maximum and minimum parameter. Standard deviation is the range of dispersion of the variables or parameter. Coefficient of skewness, a measure of the asymmetry of the probability distribution of a real -valued random variable. The skewness value can be positive or negative, or even undefined. Coefficient of kurtosis, a measure of the peakedness of the probability distribution of a real-valued random variable, although some sources are insistent that heavy tails, and not

peakedness, is what is really being measured by kurtosis. Higher kurtosis means more of the variance is the result of infrequent extreme deviations, as opposed to frequent modestly sized deviations. Normal distributions produce a kurtosis statistic of about zero. As you can see from the tables below, DENS variable is showing mean of 7904kg/m^3, standard deviation of 1422kg/m^3, -0.5120 of skewness, 0.4228 of kurtosis, 4227 as maximum and 10817 as minimum value. The values of input and output variables are shown below respectively.

Table 4.2: Statistical of Random Input Variables

Name	Mean	Standard Deviation	Skewness	Kurtosis	Maximum	Minimum
DENS	7904	1422	-0.5120	0.4228	4227	10817
ELASTIC	2.1087E+11	9.3463E+10	-0.1610	-0.2207	1.5451E+10	3.996E+11
POISON	0.2651	0.2599	-0.4257	0.6636	-0.4333	0.7715
LOAD	-5.009E+04	471.6	6.108E-2	-0.9752	-5.0913E+03	-4.93E+04
WINDLOAD	1.5007E+04	514.2	-0.1395	0.1833	1.3763E+04	1.606E+04
TEMP	66.28	43.10	-0.1583	-0.4942	-32.77	136.6

Table 4.3: Statistical of Random Output Variables

Name	Mean	Standard Deviation	Skewness	Kurtosis	Maximum	Minimum
MAXIMUMDEFLECTION	0.078	2.7E-03	-0.1395	0.1833	0.07164	0.08360
MAX_DEFLECTION	-0.01	9.6E-05	0.0639	-0.9798	-0.0106	-0.01024

4.2.3 Sample History Plots

The sample history values are a function of the number of simulation loops. The plot is only applicable for Monte Carlo Simulation results. For the mean and standard

deviation, the curves indicates that we are 95.000% sure that the "true" mean value and standard deviation are in fact between the upper and lower confidence bounds. For the minimum value, only the upper confidence bound is shown. The curve means that we are 95.000% sure that the "true" minimum value is below that curve. The same applies analogously for the maximum value, where only the lower confidence curve is shown.
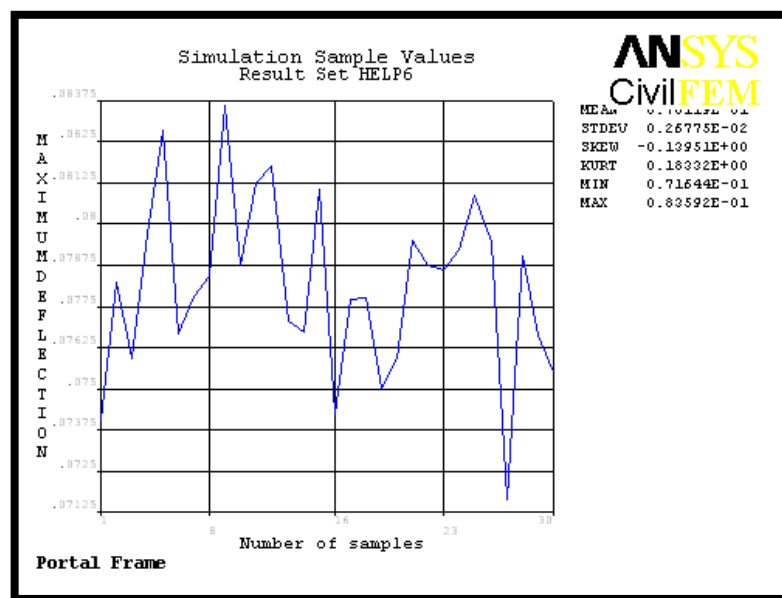
Figure 4.7: Sampled Values for Output Parameter MAXIMUMDEFLECTION

Figure 4.7 above shows the simulation sample for output parameter MAXIMUMDEFLECTION while the simulation is running for 30 samples and each sample has different MAXIMUMDEFLECTION which form the graph as shown.

Figure 4.8 is showing the mean value history for output parameter of MAXIMUMDEFLECTION. In the graph shown, we can see that the mean value of the output parameter is 0.010119. The blue line indicates the mean value while red line is indicating the limit of the mean value for the output parameter by using confidence limit of 95%.
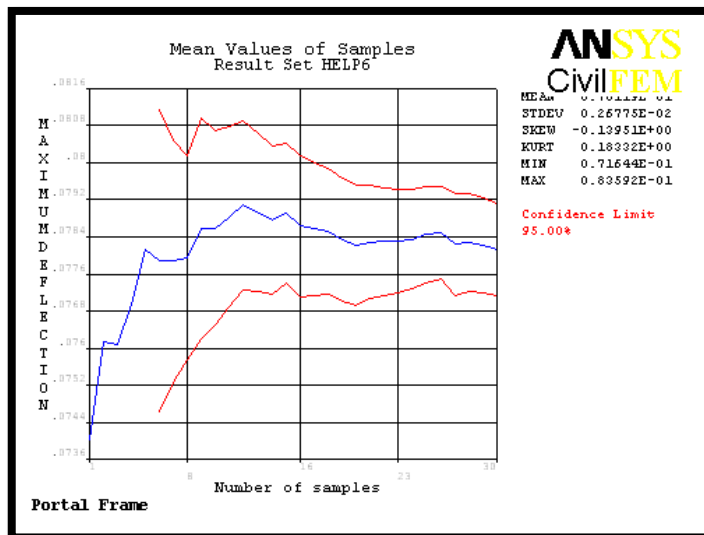
Figure 4.8: Mean Value History for Output Parameter MAXIMUMDEFLECTION

Figure 4.9 is showing the standard deviation history for output parameter of MAXIMUMDEFLECTION. In the graph shown, we can see that the standard deviation of the output parameter is 0.0026775. The blue line indicates the standard deviation while red line is indicating the limit of the standard deviation for the output parameter by using confidence limit of 95%.
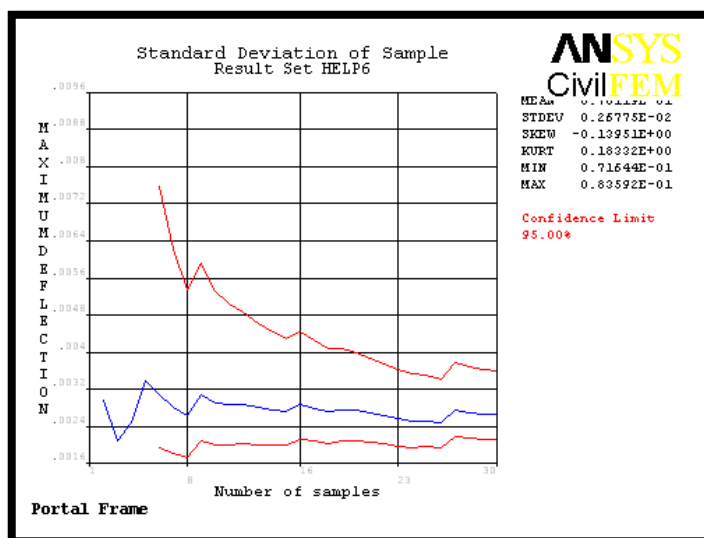
Figure 4.9: Standard Deviation History for Output Parameter
MAXIMUMDEFLECTION

Figure 4.10 is showing the minimum value for output parameter of MAXIMUMDEFLECTION. In the graph shown, we can see that the minimum value of the output parameter is 0.071644. The blue line indicates the minimum value while red line is indicating the limit of the minimum value for the output parameter by using confidence limit of 95%.
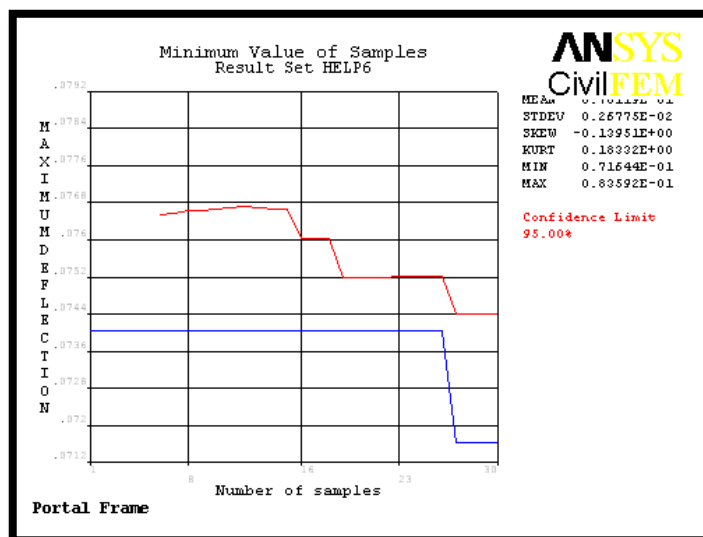
Figure 4.10: Minimum Value for Output Parameter MAXIMUMDEFLECTION
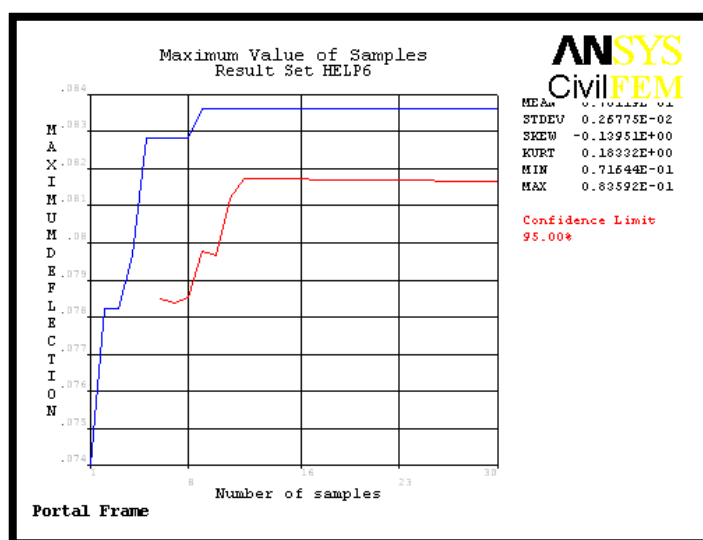
Figure 4.11: Maximum Value for Output Parameter of MAXIMUMDEFLECTION

Figure 4.11 is showing the maximum value for output parameter of MAXIMUMDEFLECTION. In the graph shown, we can see that the maximum value of the output parameter is 0.08259. The blue line indicates the maximum value while red line is indicating the limit of the maximum value for the output parameter by using confidence limit of 95%.
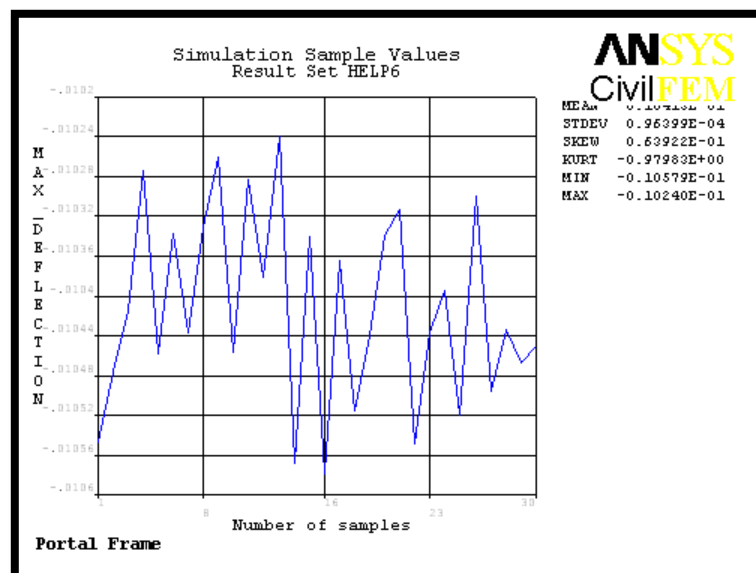
Figure 4.12: Sampled Values for Output Parameter MAX_DEFLECTION

Figure 4.12 above shows the simulation sample for output parameter MAX_DEFLECTION while the simulation is running for 30 samples and each sample has different MAXIMUMDEFLECTION which form the graph as shown.

Figure 4.13 is showing the mean value history for output parameter of MAXIMUMDEFLECTION. In the graph shown, we can see that the mean value of the output parameter is 0.010418. The blue line indicates the mean value while red line is indicating the limit of the mean value for the output parameter by using confidence limit of 95%.
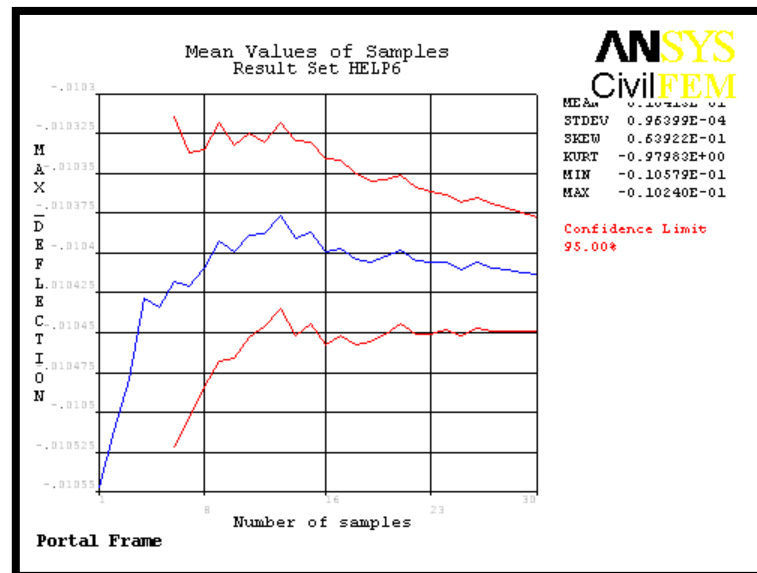
Figure 4.13: Mean Value History for Output Parameter MAX_DEFLECTION
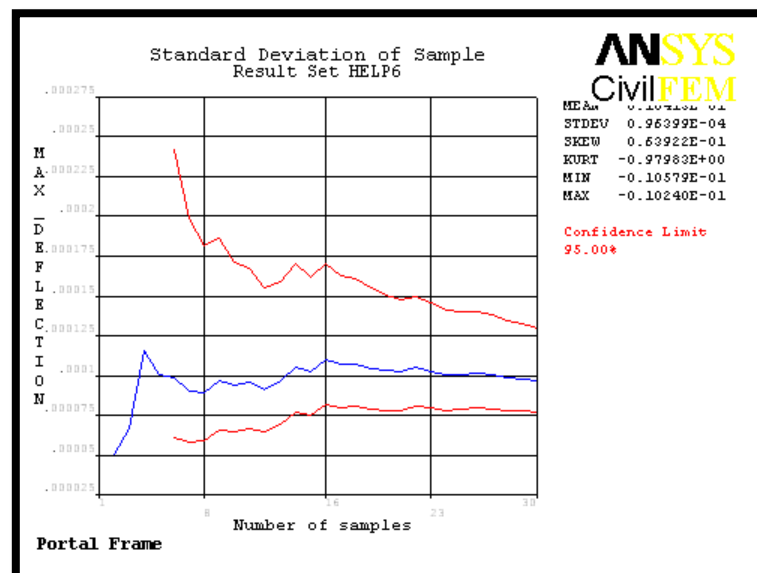
Figure 4.14: Standard Deviation History for Output Parameter MAX_DEFLECTION

Figure 4.14 is showing the standard deviation history for output parameter of MAX_DEFLECTION. In the graph shown, we can see that the standard deviation of the output parameter is 0.00096. The blue line indicates the standard deviation while red line

is indicating the limit of the standard deviation for the output parameter by using confidence limit of 95%.
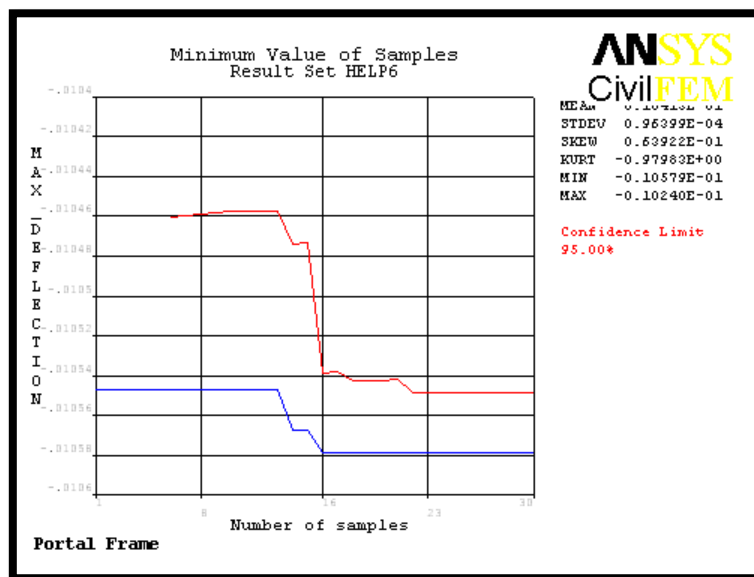
Figure 4.15: Minimum Value for Output Parameter MAX_DEFLECTION

Figure 4.15 is showing the minimum value for output parameter of MAX_DEFLECTION. In the graph shown, we can see that the minimum value of the output parameter is -0.010579. The blue line indicates the minimum value while red line is indicating the limit of the minimum value for the output parameter by using confidence limit of 95%.

Figure 4.16 is showing the maximum value for output parameter of MAX_DEFLECTION. In the graph shown, we can see that the maximum value of the output parameter is -0.01024. The blue line indicates the maximum value while red line is indicating the limit of the maximum value for the output parameter by using confidence limit of 95%.
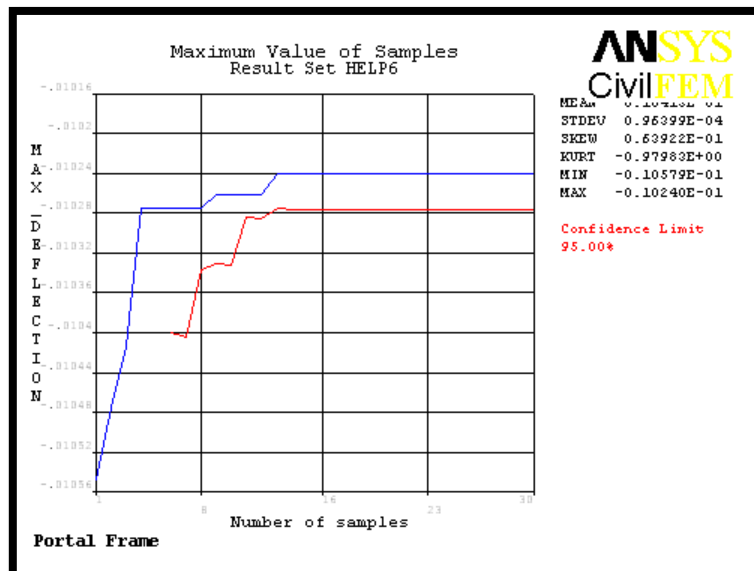
Figure 4.16: Maximum Value for Output Parameter of MAX_DEFLECTION

Figure 4.17 shows the simulation sample values for output parameter MAXIMUMDEFLECTION by 10000 numbers of sample. The X-axis is representing the number of runs in the analysis and Y axis is representing maximum defection. From the simulation results, we can see that the mean is around 0.01m. In the meanwhile, the minimum is 0.063251m and maximum is 0.089028m.
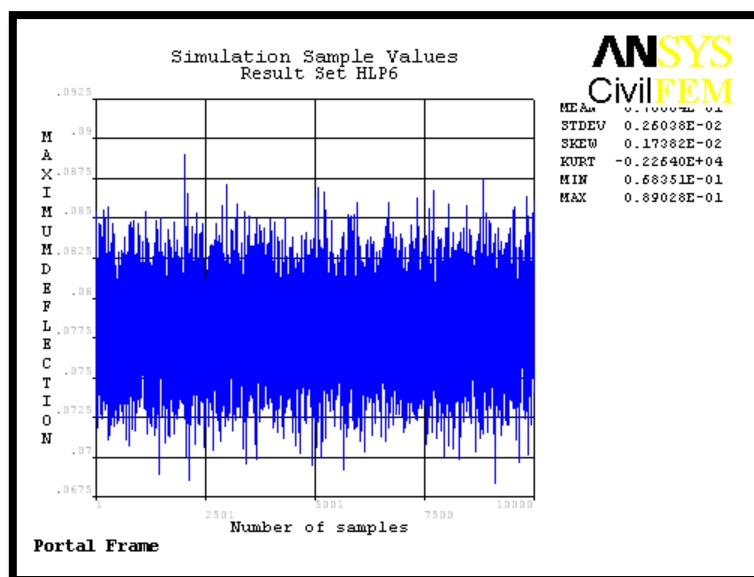
Figure 4.17: Simulation Sample Values for Output Parameter

MAXIMUMDEFLECTION
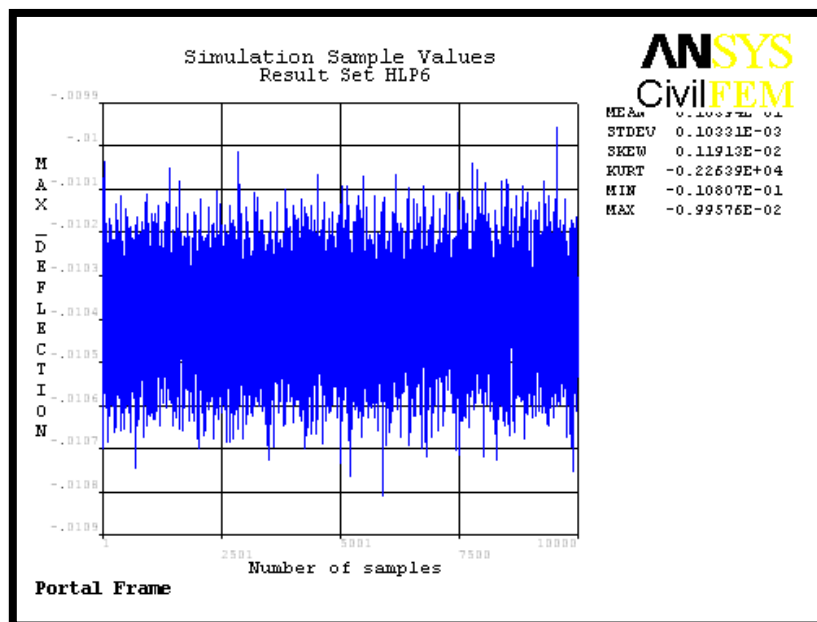
Figure 4.18: Simulation Sample Values for Output Parameter MAX_DEFLECTION

Figure 4.18 shows the simulation sample values for output parameter MAX_DEFLECTION. The X-axis is representing the number of runs in the analysis and Y axis is representing maximum defection. From what we can see from the results, it is shown that the mean value is also 0.01m. Besides that, the value for maximum is -0.099576m while for minimum value is -0.010807m.

4.2.4 Histogram Plots of Output Parameters

ANSYS PDS is able to calculate an appropriate number of classes based on the number of samples. The number of classes is equal to the number of bars shown in the histogram. Every class has the same width that gets from the divided range between the smallest and largest sample value. A histogram is obtained by the number in each individual classes and dividing this number by the total number of samples. Hence, a histogram is a graphical representative of the distribution data. Figure 4.18 and 4.19 show the histogram of output parameter for MAXIMUMDEFLECTION and MAX_DEFLECTION.
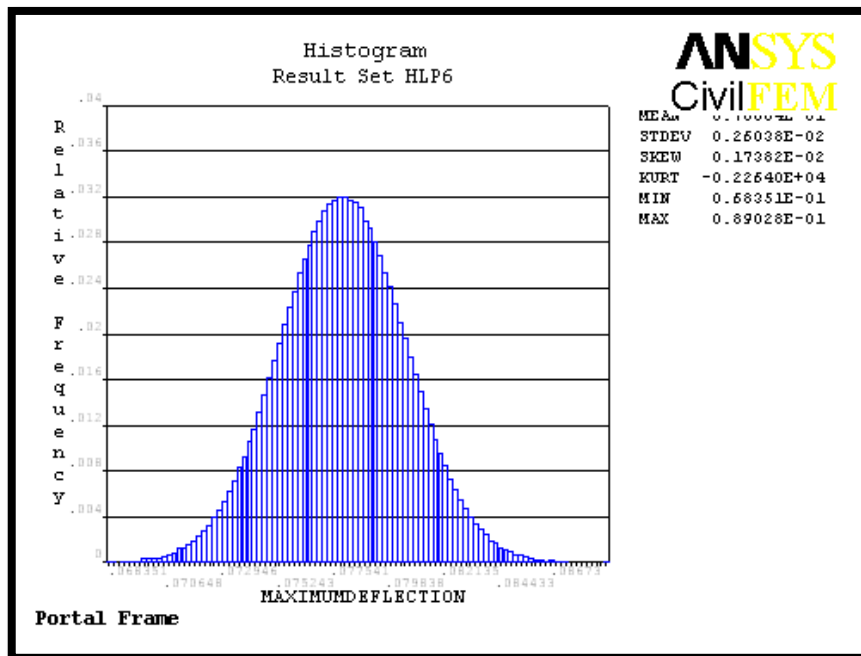
Figure 4.19: Histogram of Output Parameter for MAXIMUMDEFLECTION
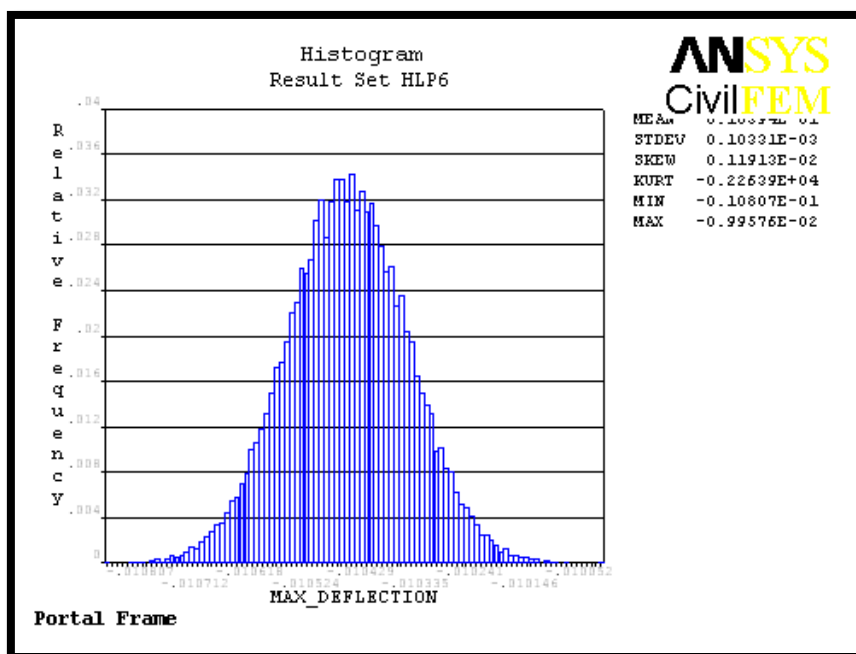
Figure 4.20: Histogram of Output Parameter for MAX_DEFLECTION

Figure 4.19 and 4.20 as shown are providing some technical statistics information on the coefficient of skewness and coefficient of kurtosis which has been broadly used in this research analysis. Coefficient of skewness, a measure of the asymmetry of the probability distribution of a real valued random variable. The skewness value can be positive or negative, or even undefined. Qualitatively, a negative skew indicates that the tail on the left side of the probability density function is longer than the right side and the bulk of the values lie to the right of the mean. A positive skew indicates that the tail on the right side is longer than the left side and bulk of the values lie to the left of the mean. A zero value indicates that the value a relatively evenly distributed on both side of the mean, typically but not necessarily implying a symmetric distribution. Normal distributions produce a skewness statistic of about zero. As what we can see from Figure 4.19 and 4.20 above, the value of skewness are 0.0017382 and 0.0011913 which are acceptable and normal because they are very close to the zero. We can conclude that they are normal distribution set of test scores and they are probably just the chances fluctuation from zero.

Coefficient of kurtosis, a measure of the peakedness of the probability distribution of a real-valued random variable, although some sources are insistent that heavy tails, and not peakedness, is what is really being measured by kurtosis. Higher kurtosis means more of the variance is the result of infrequent extreme deviations, as opposed to frequent modestly sized deviations. Normal distributions produce a kurtosis statistic of about zero. According to Figure 4.19 and 4.20 as shown above, kurtosis statistic of -0.0002264 and -0.00022639 would acceptable kurtosis value for a normal distribution because they are very close to zero.

Other than that, the minimum and maximum deflection of the model will be 68.351mm and 89.028mm according to Figure 4.19. In the meanwhile, the minimum and maximum deflection of the model will be -10.807mm and -9.9576mm according to Figure 4.20.

4.2.5   Histogram Plots of Input Parameters
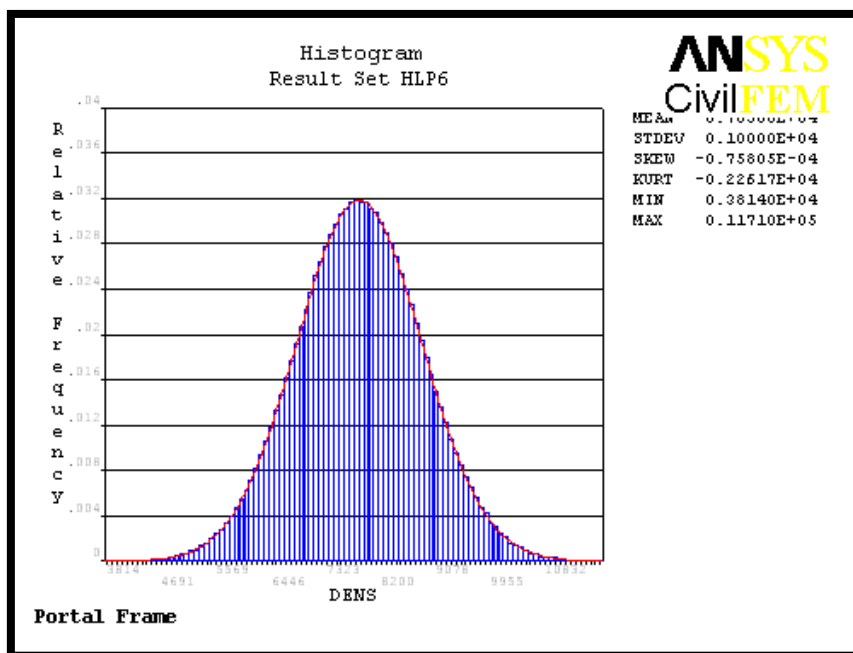
Figure 4.21: Histogram of Input Parameter for DENS

According to Figure 4.21 as shown above, we can see that the value of skewness are -0.00075805 which are acceptable and normal because they are very close to the zero. We can conclude that they are normal distribution set of test scores and they are probably just the chances fluctuation from zero. The tail on the left side is longer than right side. We can also see that the histogram has kurtosis statistic of -2261.7 as it has infrequent extreme deviations. Other than that, the minimum and maximum density of the model will be 3814kg/m^3 and 11710kg/m^3.

According to Figure 4.22 as shown below, we can see that the value of skewness are 0.00201 which are acceptable and normal because they are very close to the zero. We can conclude that they are normal distribution set of test scores and they are probably just the chances fluctuation from zero. The tail on the right side is longer than left side. We can also see that the histogram has kurtosis statistic of -2263.4 as it has infrequent extreme

deviations. Other than that, the minimum and maximum elastic modulus of the model will be -174.91GPa and 615.51Gpa.
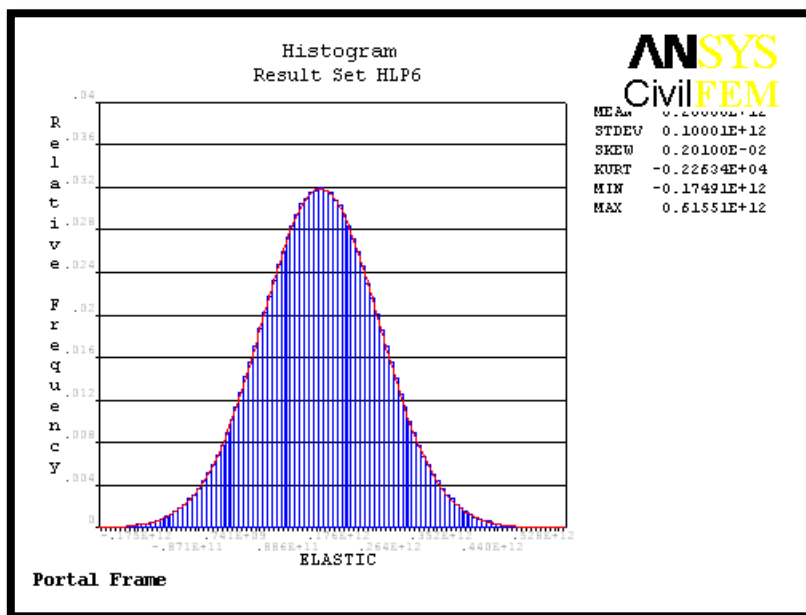
Figure 4.22: Histogram of Input Parameter for ELASTIC
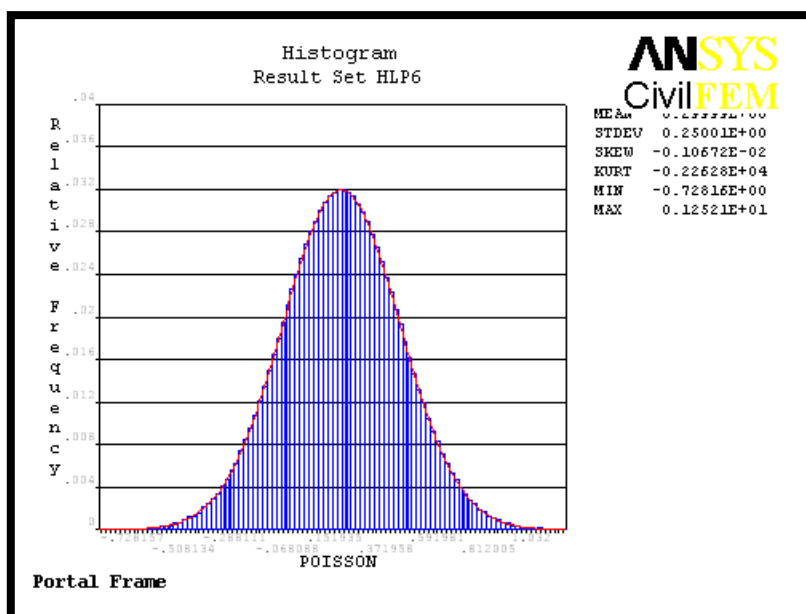
Figure 4.23: Histogram of Input Parameter for POISON

According to Figure 4.23 as shown above, we can see that the value of skewness are -0.0010672 which are acceptable and normal because they are very close to the zero. We can conclude that they are normal distribution set of test scores and they are probably just the chances fluctuation from zero. The tail on the left side is longer than right side. We can also see that the histogram has kurtosis statistic of -2262.8 as it has infrequent extreme deviations. Other than that, the minimum and maximum poison ratio will of the model be -0.72816 and 1.2521.
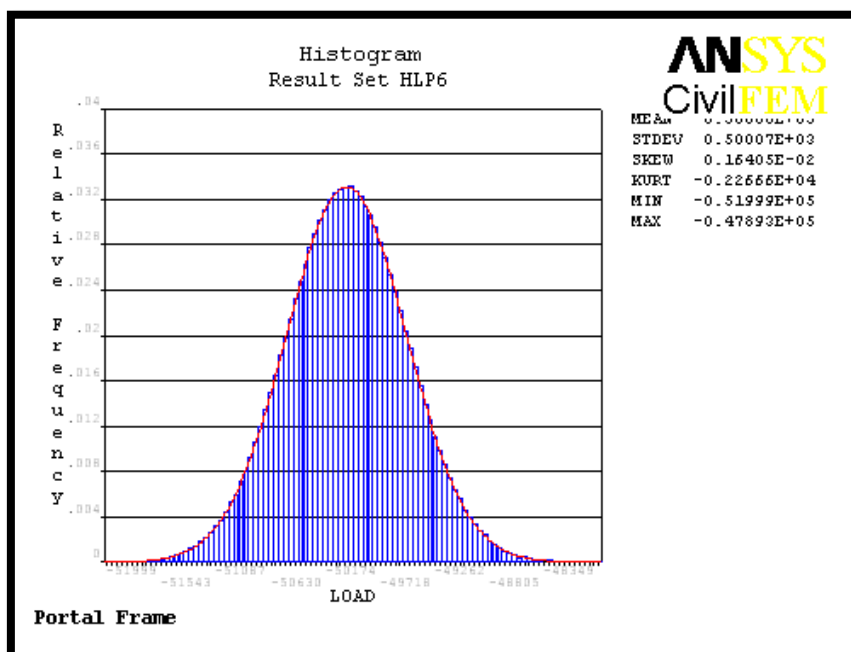
Figure 4.24: Histogram of Input Parameter for LOAD

According to Figure 4.24 as shown above, we can see that the value of skewness are 0.00164 which are acceptable and normal because they are very close to the zero. We can conclude that they are normal distribution set of test scores and they are probably just the chances fluctuation from zero. The tail on the right side is longer than left side. We can also see that the histogram has kurtosis statistic of -2255.5 as it has infrequent extreme deviations. Other than that, the minimum and maximum load of the model will be -51999kN and -47893kN.
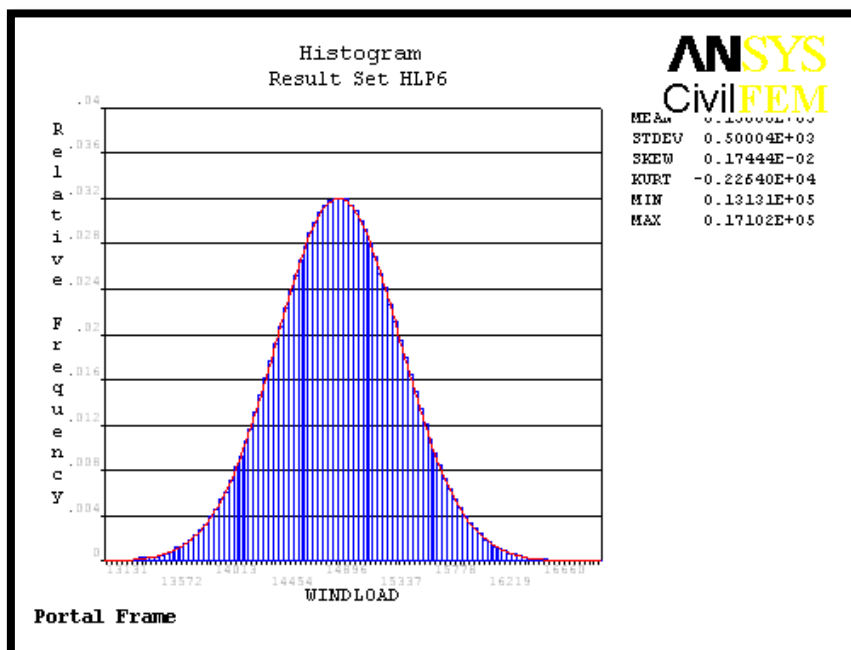
Figure 4.25: Histogram of Input Parameter for WINDLOAD

According to Figure 4.25 as shown above, we can see that the value of skewness are 0.0017444 which are acceptable and normal because they are very close to the zero. We can conclude that they are normal distribution set of test scores and they are probably just the chances fluctuation from zero. The tail on the right side is longer than left side. We can also see that the histogram has kurtosis statistic of -2264 as it has infrequent extreme deviations. Other than that, the minimum and maximum windload of the model will be 13131kN and 17102kN.

According to Figure 4.26 as shown below, we can see that the value of skewness are 0.0051523 which are acceptable and normal because they are very close to the zero. We can conclude that they are normal distribution set of test scores and they are probably just the chances fluctuation from zero. The tail on the right side is longer than left side. We can also see that the histogram has kurtosis statistic of -2278.5 as it has infrequent extreme deviations. Other than that, the minimum and maximum temperature of the model will be -82.196 degree celcius and 262.64 degree celcius.
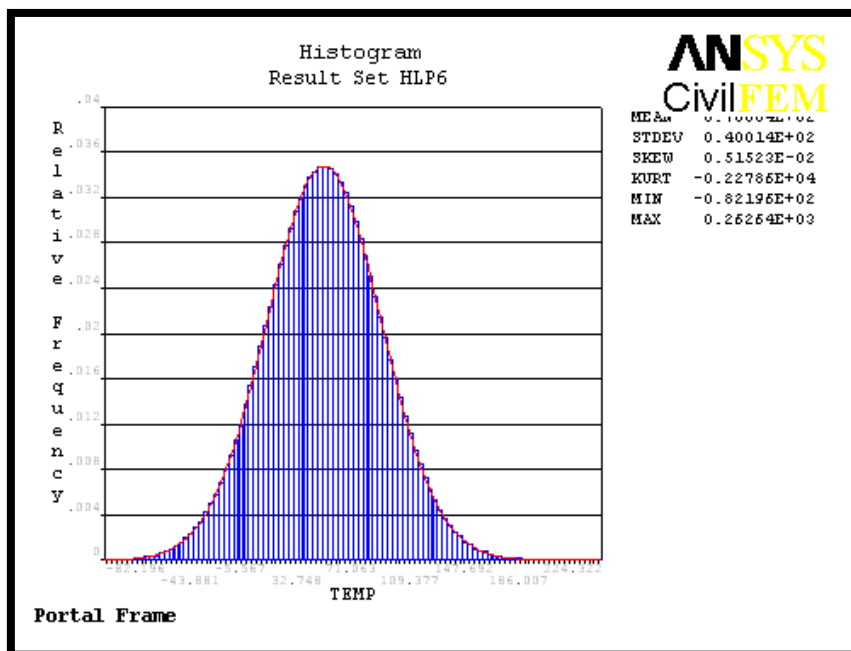
Figure 4.26: Histogram of Input Parameter for TEMP

4.2.6 Cumulative Distribution Function Plots

The cumulative distribution curve is plotted by summation of relative frequency from zero percent to hundred percent. According to Figure 4.27 to Figure 4.34 as shown below, the upper and lower curves are the confidence interval using a 95% confidence level. The confidence interval quantifies the accuracy of the probability results. After the reliability of the portal frame has been quantified, it may happen that the resulting value is not sufficient. Then, probabilistic methods can be used to answer the following question: Which input variables should be addressed to achieve a robust design and improve the quality? The answer to that question can be derived from probabilistic sensitivity diagrams plot.

For the stability of the portal frame, all the deflection value should be falling between the upper and lower curves.
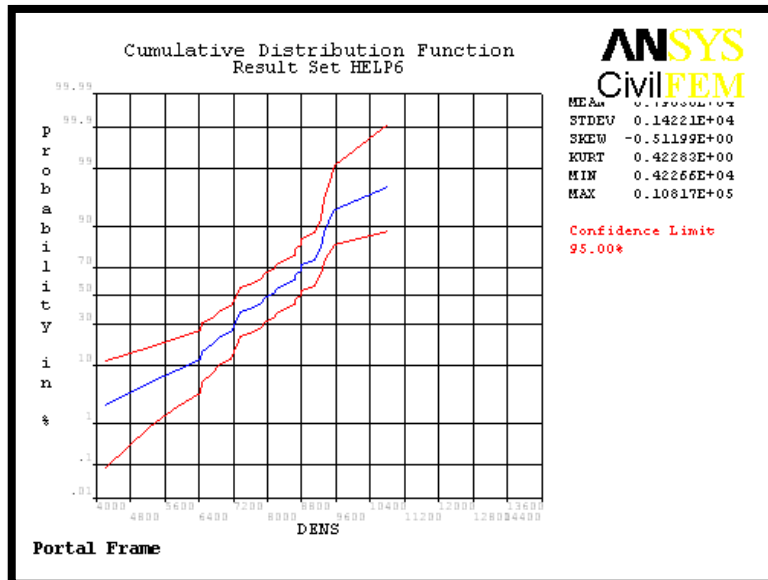
Figure 4.27: CDF of Input Variable DENS

According to Figure 4.27 as shown above, the maximum value of variable DENS should be 10817kg/m³ while the minimum should be 4226.6kg/m³. The mean value of density is falling between them which is 7850kg/m³. This shows that allow the value of density is allow to fluctuate between the range of two curves.
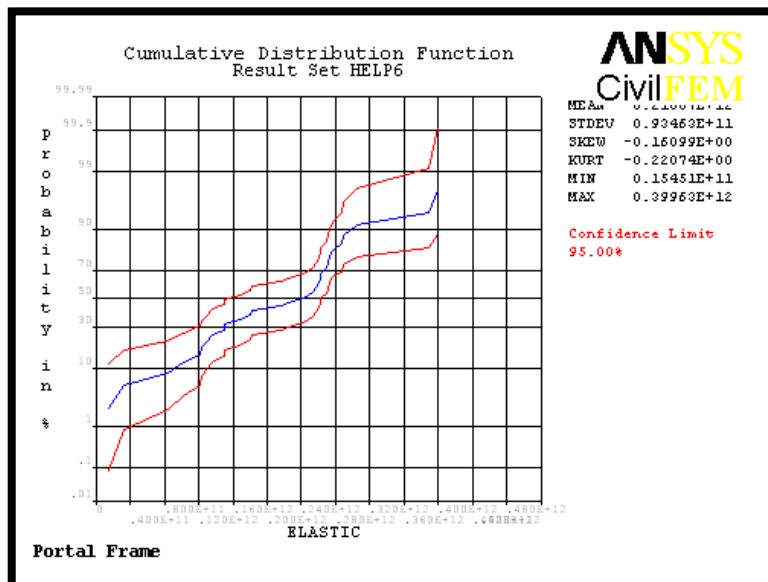
Figure 4.28: CDF of Input Variable ELASTIC

According to Figure 4.28 as shown above, the maximum value of variable ELASTIC should be 399.63GPA while the minimum should be 15.451GPA. The mean value of elastic modules is falling between them which is 210GPA. This shows that allow the value of elastic modulus is allow to fluctuate between the range of two curves.
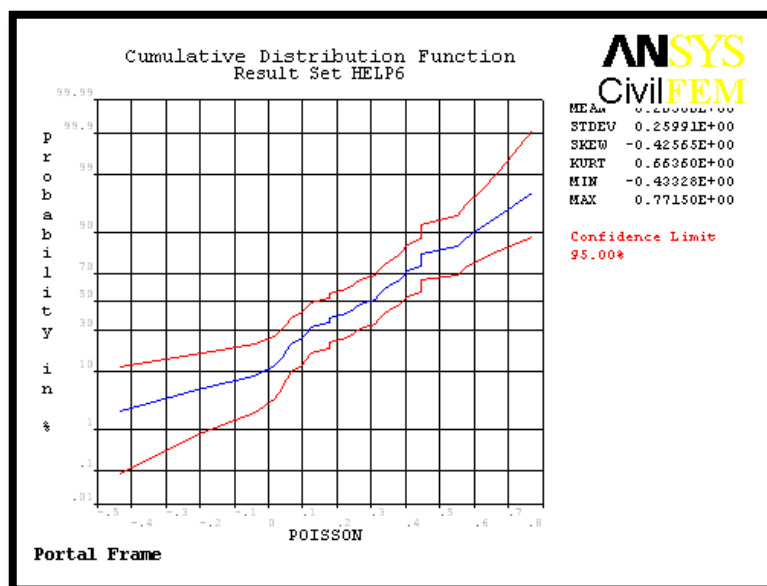
Figure 4.29: CDF of Input Variable POISON

According to Figure 4.29 as shown above, the maximum value of variable POISON should be 0.7715 while the minimum should be -0.43328. The mean value of poison ratio is falling between them which is 0.2. This shows that allow the value of poison ratio is allow to fluctuate between the range of two curves.

According to Figure 4.30 as shown below, the maximum value of variable LOAD should be -49.261kN while the minimum should be -50.913kN. The mean value of load is falling between them which is 50kN. This shows that allow the value of Load is allow to fluctuate between the range of two curves.
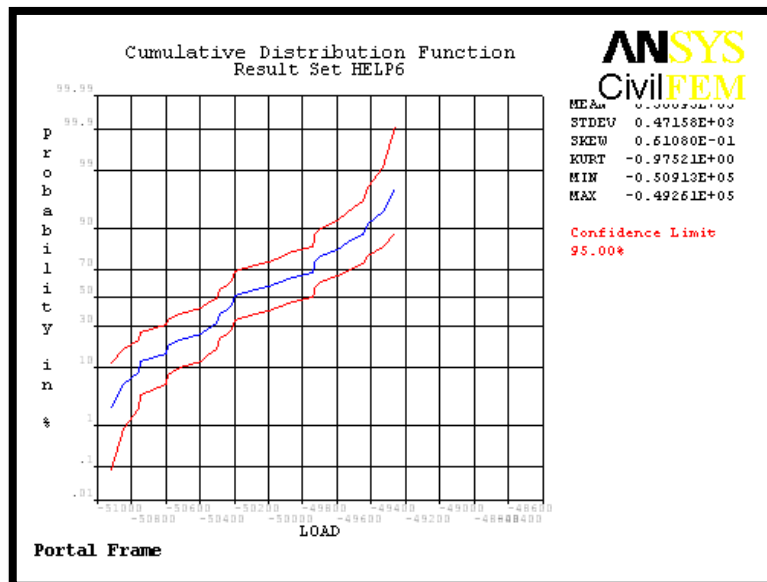
Figure 4.30: CDF of Input Variable LOAD

According to Figure 4.31 as shown below, the maximum value of variable WINDLOAD should be 16.058kN while the minimum should be 13.763kN. The mean value of wind load is falling between them which is 15kN. This shows that allow the value of Wind Load is allow to fluctuate between the range of two curves.
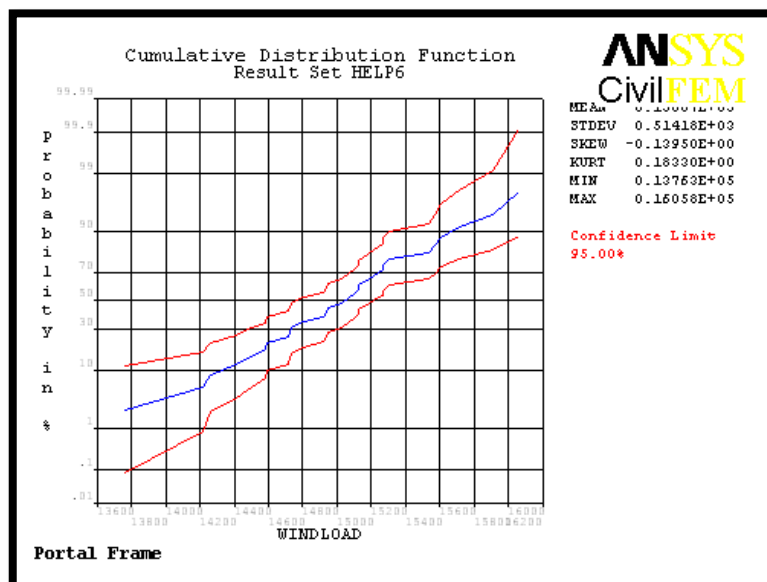
Figure 4.31: CDF of Input Variable WINDLOAD
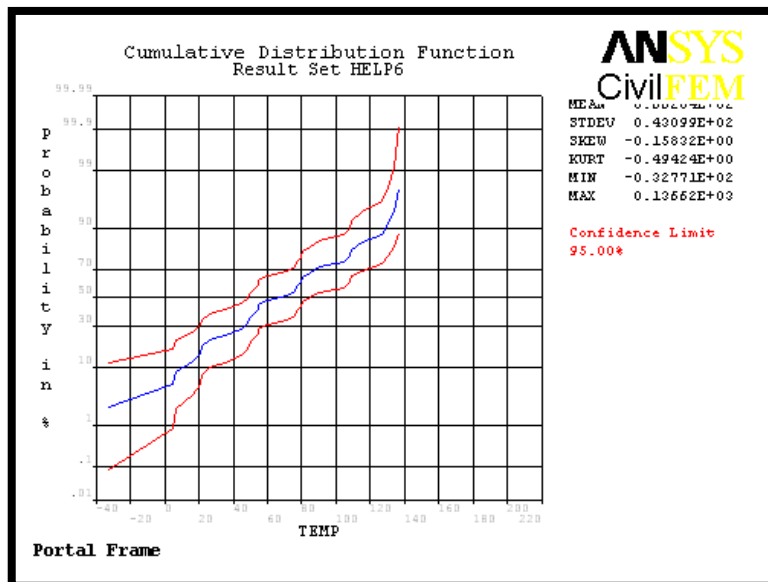
Figure 4.32: CDF of Input Variable TEMP

According to Figure 4.32 as shown above, the maximum value of variable TEMP should be 136.62 degree celcius while the minimum should be -32.77 degree celcius. The mean value of temperature is falling between them which is 70 degree celcius. This shows that allow the value of temperature is allow to fluctuate between the range of two curves.
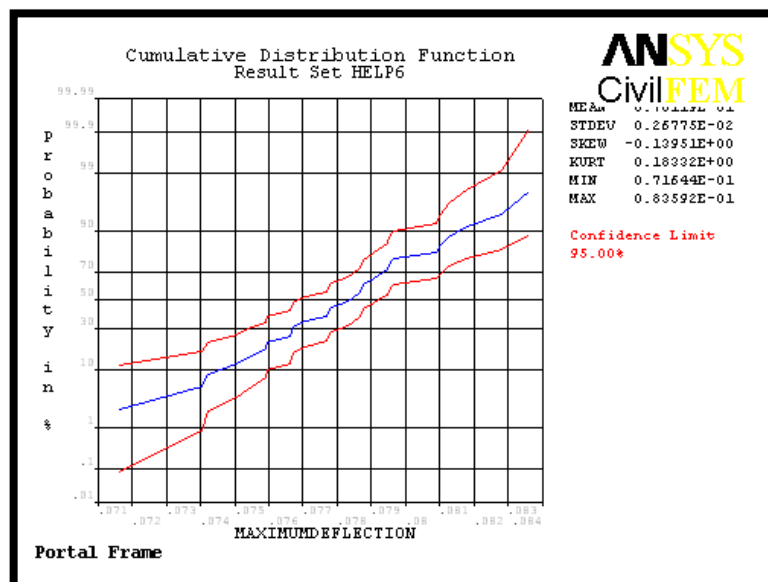
Figure 4.33: CDF of Output Variable MAXIMUMDEFLECTION

According to Figure 4.33 as shown above, the maximum value of variable MAXIMUMDEFLECTION should be 83.592mm while the minimum should be 71.644mm. This shows that allow the value of maximum deflection is allowed to fluctuate between the range of two curves.
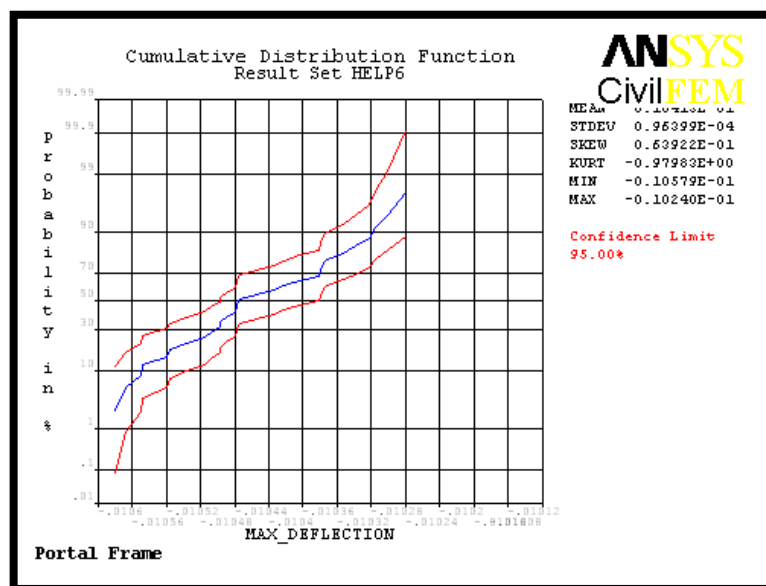
Figure 4.34: CDF of Output Variable MAX_DEFLECTION

According to Figure 4.34 as shown above, the maximum value of variable MAXI_DEFLECTION should be -10.24mm while the minimum should be -10.579mm. This shows that allow the value of maximum deflection is allowed to fluctuate between the range of two curves.

4.2.7 Sensitivity Plots

The described of probabilistic sensitivities is derived from the correlation coefficients between all the variables of LOAD, WINDLOAD, TEMP, DENS, ELASTIC and POISON and maximum deflection. In order to plot the sensitivities pie chart, the significant inputs are considered while the insignificant inputs are ignored. Significant

input means that the input is highly affecting the output values when compared to the insignificant inputs.

Figure 4.35 below shows the effect of significant random input to the MAXIMUMDEFLECTION. The deflection of the portal frame is highly affected by the input WINDLOAD and LOAD only. The other variables are insignificant. However, WINDLOAD possess the highest sensitivity to the deflection of portal according to the figure. We can roughly interpret from the figure that WINDLOAD variable is 55% significant and LOAD variable is 45% significant to the output MAXIMUMDEFLECTION while the other variables are insignificant.
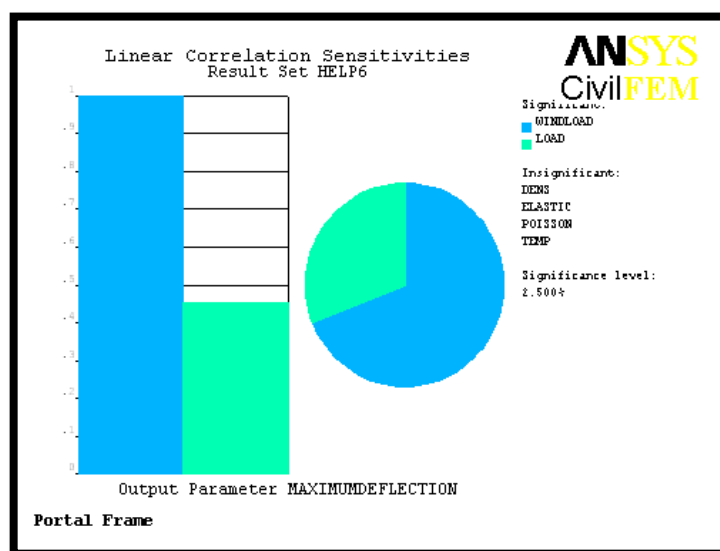
Figure 4.35: Sensitivity Plot for MAXIMUMDEFLECTION

Figure 4.36 below shows the effect of significant random input to the MAX_DEFLECTION. The output is obviously affected by LOAD, ELASTIC and WINDLOAD. The other variables are remain insignificant. However, the output is highly affected by LOAD compared to other significant variables. In that figure, we can roughly interpret that WINDLOAD variable is 4% significant, ELASTIC variable is also 4% significant and LOAD variable is 92% significant to the output MAXIMUMDEFLECTION while the other variables are insignificant.
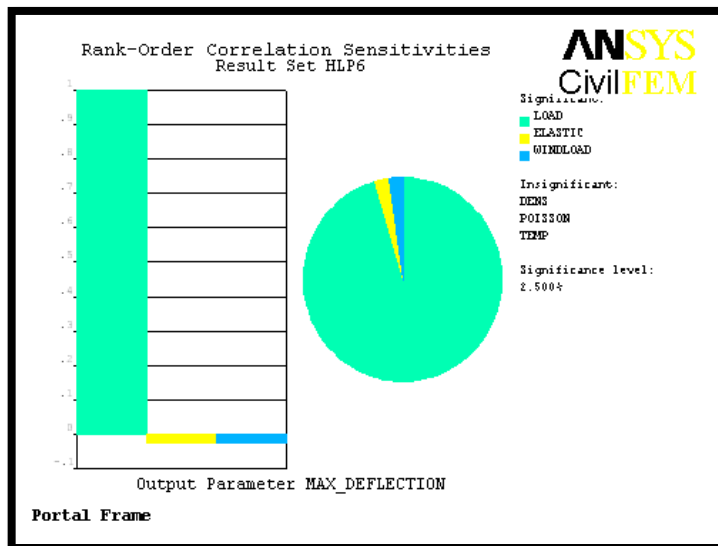
Figure 4.36: Sensitivity Plot for MAX_DEFLECTION

4.2.8   Linear Correlation Coefficients

Table 4.4, 4.5 and 4.6 are showing the linear correlation coefficients between input variables, between input and output variables and between output variables. In the tables, we can see that there is positive, negative and 1 as the value of the coefficient. The positive values are indication that the linear relationship is increasing which means perfect direct relationship. However, negative value is indicating decreasing linear relationship or we can call it as anticorrelation. The values are indicating the degree of linear dependence between the variables. As it approaches 1, it means that the correlation between the variables, vice versa. As we can see from the table, variables of DENS/DENS, ELASTIC/ELASTIC and so on are indicating 1 as they are linearly dependence. POISON/WINDLOAD is indicating the lowest positive value of 0.026 which means least increase linear relationship while TEMP/LOAD and TEMP/ELASTIC is showing the lowest negative value of -0.093 which means least decrease liner relationship.

Table 4.4: Linear Correlation Coefficients between Input Variables

Inp/Inp	DENS	ELASTIC	POISON	LOAD	WINDLOAD	TEMP
DENS	1.000	-0.130	0.162	0.126	0.056	0.283
ELASTIC	-0.130	1.000	0.304	0.369	0.044	-0.105
POISON	0.162	0.304	1.000	0.278	0.026	0.157
LOAD	0.126	0.369	0.278	1.000	0.451	-0.093
WINDLOAD	0.056	0.044	0.026	0.451	1.000	0.163
TEMP	0.283	-0.102	0.157	-0.093	0.163	1.000

Table 4.5: Linear Correlation Coefficients between Input and Output Variables

Out/Inp	DENS	ELASTIC	POISON	LOAD	WINDLOAD	TEMP
MAXIMUMDEFLECTION	0.056	0.044	0.026	0.451	1.000	0.163
MAX_DEFLECTION	0.126	0.372	0.280	1.000	0.432	-0.098

Table 4.6: Linear Correlation Coefficients between Output Variables

Out/Out	MAXIMUMDEFLECTION	MAX_DEFLECTION
MAXIMUMDEFLECTION	1.000	0.432
MAX_DEFLECTION	0.432	1.000

4.2.9   Spearman Rank Order Correlation Coefficients

Spearman's rank correlation coefficient is a nonparametric measure of statistical dependence between two variables by describing it using monotonic function. The sign

of the Spearman correlation indicates the direction of the association between X (independent variable) and Y (dependent variable). According to Table 4.7, 4.8 and 4.9 that show the spearman rank order correlation coefficients between input variables, between input and output variables and between output variables, we can see that there are two types of sign which are negative and positive. Positive value indicates the Y tends to increase when X increases while negative value indicates the Y tends to decrease when X increases. Spearman correlation of +1 or -1 are indicating that the correlation is perfect. In the tables below, we can see that the highest value for positive value is 0.450 by LOAD/WINDLOAD and the highest for negative value is -0.126 by ELASTIC/DENS which indicating the strongest association between X and Y variables in different directions.

Table 4.7: Spearman Rank Order Correlation Coefficients between Input Variables

Inp/Inp	DENS	ELASTIC	POISON	LOAD	WINDLOAD	TEMP
DENS	1.000	-0.126	0.150	0.119	0.030	0.318
ELASTIC	-0.126	1.000	0.367	0.308	0.028	-0.099
POISON	0.150	0.367	1.000	0.301	0.044	0.063
LOAD	0.119	0.308	0.301	1.000	0.450	-0.106
WINDLOAD	0.030	0.028	0.044	0.450	1.000	0.164
TEMP	0.318	-0.099	0.063	-0.106	0.164	1.000

Table 4.8: Spearman Rank Order Correlation Coefficients between Input and Output Variables

Out/Inp	DENS	ELASTIC	POISON	LOAD	WINDLOAD	TEMP
MAXIMUMDEFLECTION	0.030	0.028	0.044	0.450	1.000	0.164
MAX_DEFLECTION	0.0123	0.29	0.311	0.998	0.422	-0.110

Table 4.9: Spearman Rank Order Correlation Coefficients between Output Variables

Out/Out	MAXIMUMDEFLECTION	MAX_DEFLECTION
MAXIMUMDEFLECTION	1.000	0.422
MAX_DEFLECTION	0.422	1.000

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1   INTRODUCTION

In this chapter, all the results that obtained from ANSYS and manual calculations were summarized and discussed. Other than that, there are some suggestions are made in this chapter in order to improve the future research study.

5.2   CONCLUSION

According the results generated from the ANSYS about 3D steel portal frame structure design, it is obviously shown that the steel structure was satisfied in the geometry and material used. The portal frame was satisfied in term of everything by checking it accordingly to Eurocode 3 by manual calculation. In compared to ANSYS results, the figures shown in Chapter 3 are also showing green colour which means pass.

The general results of displacement and element forces were analyzed in ANSYS and compared with manual calculation. The results shown are almost similar. In the meanwhile, the sample history plots, histogram plots, cumulative distribution function

plots and sensitivity plots for the general results were determined by using probabilistic method with random input of LOAD, WINDLOAD, TEMP, DENS, ELASTIC and POISON.

In conclusion, not all the inputs are affecting the results of the analysis. The inputs that affecting the results are called significant inputs. It is very important to carry out the research by identifying more significant inputs for this research. This is because the sample of probabilistic study on 3D portal frame is very limited. Therefore, more research with different significant input should be carried out.

5.3   RECOMMENDATION

This research has been done accordingly to the requirement. However, there are some improvements can be made in order to increase the accuracy of this results. There are some ways to improve and gather more information for the structure analysis and checking. The recommendations are as shown below:

(i)   More significant inputs to be provided

(ii)   More detail design of the structure to be tested

(iii)   Different shape of cross section to be tested

REFERENCES

BS EN 1991-1-1:. (2002). *Eurocode 1: Actions on structures. General actions. Densities, self-weight, imposed loads for buildings , BSI.*

Davidson, B., & W. Owens, G. (2012). *STEEL DESIGNERS' MANUAL.* BLACKWELL PUBLISHING LTD.

Duoc, T., James B.P., L., Tiku T. , T., R. Mark, L., Yixiang, X., Steven, M., & Wei, S. (2013). Effect of serviceability limits on optimal design of steel portal frames. *Journal of Constructional Steel Research*, 74-84.

Elsayed, M., Mohamed, E.-H., Hamdy, A.-E., & Mohamed, O. (2010). Finite element analysis of beam-to-column joints. *Alexandria Engineering Journal*, 91-104.

G Lackshmi, N. (2009). *Finite Element Analysis.* BS Publications.

M.T., R.-L., & Jose, S.-S. (2014). Analysis of wind action on unique structures with application to Seville. *Engineering Structure*.

Mohamed, H.-Z., David, W., & M., A. (1983). NONLINEAR FINITE ELEMENT ANALYSIS. *J. Struct. Eng.*, 353-368.

P.J., M., R.P., D., M.W. , B., & A.H. , B. (2008). Design of steel portal frame buildings for fire safety. *Journal of Constructional Steel Research*, 1216-1224.

Ross, M., James, B., Tiku, T., Duoc, T., & Wei Sha. (2014). Optimal design of long-span steel portal frames using fabricated beams. *Journal of Constructional Steel Research*, 104-114.

Steel Construction. (2014). Retrieved from Steel Construction: http://www.steelconstruction.info/Portal_frames

University Deakin. (2004). Retrieved from Frame Construction: http://www.ab.deakin.edu.au/online/vgallery/2004/srt251/team22/Home/navigation/Framing/Frame.htm#top

University of Alberta - ANSYS Tutorials. (2001). Retrieved from http://www.mece.ualberta.ca/tutorials/ansys/

APPENDIX A1

EUROCODE 3 CHECKING OF PORTAL FRAME

Structure Properties

Spacing of portal frame = 5m

Spacing between purlins = 1.5m

Member Properties

Cross section: *IPE A 600*

Material: *Fe E355*

Section Properties

$h = 597$mm $\qquad\qquad\qquad A = 13700\text{mm}^2$

$b = 220$mm $\qquad\qquad\qquad I_y = 118 \text{ x } 10^4\text{mm}^4$

$t_w = 9.8$mm $\qquad\qquad\qquad I_z = 3116 \text{ x } 10^4\text{mm}^4$

$t_f = 17.5$mm $\qquad\qquad\qquad W_{pl,y} = 442.1 \text{ x } 10^3\text{mm}^3$

$r = 24$mm $\qquad\qquad\qquad i_z = 43.1$mm

$h_w = 562$mm $\qquad\qquad\qquad I_w = 1249 \text{ x } 10^9\text{mm}^6$

$d = 514$mm

Design of Column

Height of column = 11m

N_{Ed} = 140kN

V_{Ed} = 16kN

M_{Ed} = 17kNm

Cross Section Classification

Web :

d/t_w = 514/9.8 = 52.45

$d_N = N_{Ed}/t_w f_y$ = 140000/(9.8x355) = 40.24

$\alpha = (d_w\ d_N)/(2d_N)$ = (514+40.24)/(2x514) = 0.54 $\geq$ 0.5

Limit for Class 1 :

$369\varepsilon/(13\alpha\text{-}1)$ = (369x0.81)/(13x0.54-1) = 53.28 > 52.45

Therefore, section can be classified as class 1.

Flange :

c/t_f = 73.9/17.5 = 4.22

Limit for Class 1 = 9ε

ε = 0.81, 9ε = 7.29

$9\varepsilon > c/t_f$

Therefore, section can be classified as class 1.

Resistance of The Cross Section

Shear Resistance:

$A_v = A\text{-}2bt_f + (t_w + 2r)t_f = 13700\text{-}(2\text{x}220\text{x}17.5) + [9.2 + 2(24)]17.5 = 7011.5\text{mm}^2$

$\eta h_w t_w = 1.0\text{x}562\text{x}9.8 = 5507.6\text{mm}^2 < A_v = 7011.5\text{mm}^2$

$V_{pl,Rd} = [A_v(f_y/\sqrt{3})/\gamma_{M0}] = [7011.5(355/\sqrt{3})/1.0]\text{x}10^{-3} = 1437.1\text{kN} > V_{Ed} = 16\text{kN}$

Therefore, the shear resistance checking is satisfactory.

Bending & Shear Interaction:

$V_{Ed} = 16\text{kN} < 0.5V_{pl,Rd} = 0.5(1437.1) = 718.55\text{kN}$

Therefore, the Bending & Shear checking is satisfactory.

Compression Resistance:

$N_{c,Rd} = (Af_y)/\gamma_{M0} = (13700\text{x}355)\text{x}10^{-3} = 4863.5\text{kN}$

$N_{Ed} = 140\text{kN} < N_{c,Rd} = 4863.5\text{kN}$

Therefore, the Compression Resistance checking is satisfactory.

Bending & Axial Force Interaction:

$0.25N_{pl,Rd} = 0.25\text{x}4863.5 = 1215.88\text{kN}$

$(0.5h_w t_w f_y)/\gamma_{M0} = [(0.5\text{x}562\text{x}9.8\text{x}355)/1.0]\text{x}10^{-3} = 977.6\text{kN}$

$N_{Ed} = 140\text{kN} < 0.25N_{pl,Rd} < (0.5h_w t_w f_y)/\gamma_{M0}$

Therefore, the Bending & Axial Force Interaction checking is satisfactory.

Bending Moment Resistance:

$M_{pl,Rd} = (w_{py}f_y)/\gamma_{M0} = [(442.1\text{x}10^{-3}\text{x}355)/1.0]\text{ x}10^{-6} = 156.95\text{kNm} > M_{Ed} = 17\text{kNm}$

Therefore, the Bending Moment Resistance checking is satisfactory.

Out-of-Plane Buckling

Verification of spacing between intermediate restraints

$C_1 = 1.77$

$$L_m = \frac{38i_z}{\sqrt{\frac{1}{57.4}\left(\frac{N_{Ed}}{A}\right)+\frac{1}{756C_1^2}\frac{W_{pl,y}^2}{AI_t}\left(\frac{f_y}{235}\right)^2}} = \frac{38\times43.1}{\sqrt{\frac{1}{57.4}\left(\frac{140000}{13700}\right)+\frac{1}{756\times1.77^2}\frac{442100^2}{13700\times89300}\left(\frac{355}{235}\right)^2}} = 1698.03\text{mm}$$

Side rail spacing = $2750 > L_m = 1698.03\text{mm}$

Whole Column (11000mm):

$h/b = 597/220 = 2.71$

$t_f = 17.5\text{mm}$

curve b

$\alpha = 0.34$

$\lambda_1 = 93.9\varepsilon = 93.9\text{x}0.81 = 76.06$

$\lambda_z = \frac{L_{cr}}{i_z}\frac{1}{\lambda_1} = (11000/43.1)\text{x}(1/76.06) = 3.36$

$\emptyset_z = 0.5[1+\alpha_z(\lambda_z\text{-}0.2)\ \lambda_z^2] = 0.5[1+0.34(3.36\text{-}0.2)+3.36^2] = 6.68$

$\chi_z = \frac{1}{\emptyset z+\sqrt{\emptyset_z^2-\lambda_z^2}} = \frac{1}{6.68+\sqrt{6.68^2-3.36^2}} = 0.08$

$N_{b,z,Rd} = \chi_z Af_y/\gamma_{M1} = [(0.08\text{x}13700\text{x}355)/1.0]\text{ x}10^{-3} = 389\text{kN} > N_{Ed} = 140\text{kN}$

Therefore, the checking is satisfactory.

Lateral Torsional Buckling Resistance, $M_{b,Rd}$:

$C_1 = 1.77$

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{L^2} \sqrt{\frac{I_w}{I_z} + \frac{L^2 GI_t}{\pi^2 EI_z}} = 1.77 \frac{\pi^2 \times 210000 \times 3116 \times 10^4}{11000^2} \sqrt{\frac{1249 \times 10^9}{3116 \times 10^4} + \frac{11000^2 \times 81000 \times 89.3 \times 10^4}{\pi^2 \times 210000 \times 3116 \times 10^4}}$$

$= 1108.8 \text{kNm}$

$$\lambda_{LT} = \sqrt{\frac{W_y f_y}{M_{cr}}} = \sqrt{\frac{442.1 \times 10^3 \times 355}{1108.8 \times 10^6}} = 0.38$$

$\lambda_{LT,0} = 0.4$

$\beta = 0.75$

$h/b = 597/220 = 2.71$

curve c

$\lambda_{LT} = \frac{L_{cr}}{i_z} \frac{1}{\lambda 1} = (11000/43.1) \times (1/76.06) = 3.36$

$\emptyset_{LT} = 0.5[1 + \alpha_{LT}(\lambda_{LT} - \lambda_{LT,0}) \beta \lambda_{LT}^2] = 0.5[1 + 0.49(0.38 - 0.4) + (0.75 \times 0.38^2)] = 0.55$

$$\chi_{LT} = \frac{1}{\emptyset_{LT} + \sqrt{\emptyset_{LT}^2 - \beta \lambda_{LT}^2}} = \frac{1}{0.55 + \sqrt{0.55^2 - 0.75 \times 0.38^2}} = 1$$

$$M_{b,Rd} = \frac{\chi_{LT} W_{pl,y} f_y}{\gamma_{M1}} = \frac{1.0 \times 442.1 \times 10^3 \times 355}{1.0} \text{ x}10^{-6} = 156.95 \text{kNm} > M_{Ed} = 17 \text{kNm}$$

Therefore, the checking is satisfactory.

Interaction of Axial Force and Bending Moment:

$\lambda_z = 3.36 > 0.4$

$\psi = 0$

$C_{mLT} = 0.6 + 0.4\psi = 0.6$

$$K_{zy} = max\left[\left(1 - \frac{0.1\lambda_z}{C_{mLT} - 0.25} \frac{N_{Ed}}{N_{b,Rd,z}}\right); \left(1 - \frac{0.1}{C_{mLT} - 0.25} \frac{N_{Ed}}{N_{b,Rd,z}}\right)\right]$$

$$K_{zy} = max\left[\left(1 - \frac{0.1 \times 3.36}{0.6 - 0.25} \frac{140}{389}\right); \left(1 - \frac{0.1}{0.6 - 0.25} \frac{140}{389}\right)\right] = max[0.654; 0.897] = 0.897$$

$$\frac{N_{Ed}}{N_{b,z,Rd}} + k_{zy}\frac{M_{y,Ed}}{M_{b,Rd}} \le 1.0$$

$$\frac{140}{389} + 0.897\,\frac{17}{165.95} \le 1.0 = 0.45 < 1.0$$

Therefore, the checking is satisfactory.

In Plane Buckling

$h/b = 597/220 = 2.71$

$t_f = 17.5$mm

buckling about y axis

curve a

$\alpha = 0.21$

$\lambda_1 = 93.9\varepsilon = 93.9x0.81 = 76.06$

$\lambda_y = \frac{L_{cr}}{i_y}\frac{1}{\lambda1} = (11000/43.1)x(1/76.06) = 3.36$

$\emptyset_y = 0.5[1+\alpha_y(\lambda_y\text{-}0.2)\;\lambda_y^2] = 0.5[1+0.21(3.36\text{-}0.2)+3.36^2] = 6.48$

$\chi_y = \dfrac{1}{\emptyset y+\sqrt{\emptyset_y^2-\lambda_y^2}} = \dfrac{1}{6.48+\sqrt{6.48^2-3.36^2}} = 0.08$

$N_{b,y,Rd} = \chi_y A f_y/\gamma_{M1} = [(0.08x13700x355)/1.0]\ x10^{-3} = 389$kN $> N_{Ed} = 140$kN

Therefore, the checking is satisfactory.

Lateral Torsional Buckling Resistance, $M_{b,Rd}$:

$C_1 = 1.77$

$$M_{cr} = C_1\,\frac{\pi^2 EI_z}{L^2}\sqrt{\frac{I_w}{I_y} + \frac{L^2 GI_t}{\pi^2 EI_y}} = 1.77\frac{\pi^2\times210000\times3116\times10^4}{11000^2}\sqrt{\frac{1249\times10^9}{3116\times10^4} + \frac{11000^2\times81000\times89.3\times10^4}{\pi^2\times210000\times3116\times10^4}}$$

$$= 1108.8\text{kNm}$$

$$\lambda_{LT} = \sqrt{\frac{W_y f_y}{M_{cr}}} = \sqrt{\frac{442.1 \times 10^3 \times 355}{1108.8 \times 10^6}} = 0.38$$

$\lambda_{LT,0} = 0.4$

$\beta = 0.75$

$h/b = 597/220 = 2.71$

curve c

$$\lambda_{LT} = \frac{L_{cr}}{i_y} \frac{1}{\lambda 1} = (11000/43.1) \times (1/76.06) = 3.36$$

$\emptyset_{LT} = 0.5[1 + \alpha_{LT}(\lambda_{LT} - \lambda_{LT,0})\beta\lambda_{LT}^2] = 0.5[1 + 0.49(0.38 - 0.4) + (0.75 \times 0.38^2)] = 0.55$

$$\chi_{LT} = \frac{1}{\emptyset_{LT} + \sqrt{\emptyset_{LT}^2 - \beta\lambda_{LT}^2}} = \frac{1}{0.55 + \sqrt{0.55^2 - 0.75 \times 0.38^2}} = 1$$

$$M_{b,Rd} = \frac{\chi_{LT} W_{pl,y} f_y}{\gamma_{M1}} = \frac{1.0 \times 442.1 \times 10^3 \times 355}{1.0} \text{ x} 10^{-6} = 156.95\text{kNm} > M_{Ed} = 17\text{kNm}$$

Therefore, the checking is satisfactory.

Interaction of Axial Force and Bending Moment:

$\lambda_y = 3.36 > 0.4$

$\psi = 0$

$C_{mLT} = 0.6 + 0.4\psi = 0.6$

$$K_{zy} = \min\left[\left(C_{my}\left(1 + \left(\lambda_y - 0.2\right)\frac{N_{Ed}}{N_{b,y,Rd}}\right)\right); \left(C_{my}\left(1 + 0.8\frac{N_{Ed}}{N_{b,y,Rd}}\right)\right)\right]$$

$$K_{zy} = \min\left[\left(0.6\left(1 + (3.36 - 0.2)\frac{140}{389}\right)\right); \left(0.6\left(1 + 0.8\frac{140}{389}\right)\right)\right] = \min[0.89; 0.77] = 0.77$$

$$\frac{N_{Ed}}{N_{b,z,Rd}} + k_{zy}\frac{M_{y,Ed}}{M_{b,Rd}} \leq 1.0$$

$$\frac{140}{389} + 0.77\frac{17}{165.95} \leq 1.0 = 0.44 < 1.0$$

Therefore, the checking is satisfactory.

Design of Rafter

Length of Rafter = 6.08m

N_{Ed} = 53kN

M_{Ed} = 2kNm

<u>Cross Section Classification</u>

Web :

d/t_w = 514/9.8 = 52.45

$d_N = N_{Ed}/t_w f_y$ = 140000/(9.8x355) = 40.24

$\alpha = (d_w\ d_N)/(2d_N)$ = (514+40.24)/(2x514) = 0.54 ≥ 0.5

Limit for Class 1 :

$369\varepsilon/(13\alpha-1)$ = (369x0.81)/(13x0.54-1) = 53.28 > 52.45

Therefore, section can be classified as class 1.

Flange :

c/t_f = 73.9/17.5 = 4.22

Limit for Class 1 = 9ε

ε = 0.81, 9ε = 7.29

$9\varepsilon > c/t_f$

Therefore, section can be classified as class 1.

Resistance of The Cross Section

Shear Resistance:

$A_v = A\text{-}2bt_f + (t_w + 2r)t_f = 13700\text{-}(2\text{x}220\text{x}17.5) + [9.2 + 2(24)]17.5 = 7011.5\text{mm}^2$

$\eta h_w t_w = 1.0\text{x}562\text{x}9.8 = 5507.6\text{mm}^2 < A_v = 7011.5\text{mm}^2$

$V_{pl,Rd} = [A_v(f_y/\sqrt{3})/\gamma_{M0}] = [7011.5(355/\sqrt{3})/1.0]\text{x}10^{-3} = 1437.1\text{kN} > V_{Ed} = 16\text{kN}$

Therefore, the shear resistance checking is satisfactory.

Bending & Shear Interaction:

$V_{Ed} = 16\text{kN} < 0.5V_{pl,Rd} = 0.5(1437.1) = 718.55\text{kN}$

Therefore, the Bending & Shear checking is satisfactory.

Compression Resistance:

$N_{c,Rd} = (Af_y)/\gamma_{M0} = (13700\text{x}355)\text{x}10^{-3} = 4863.5\text{kN}$

$N_{Ed} = 140\text{kN} < N_{c,Rd} = 4863.5\text{kN}$

Therefore, the Compression Resistance checking is satisfactory.

Bending & Axial Force Interaction:

$0.25N_{pl,Rd} = 0.25\text{x}4863.5 = 1215.88\text{kN}$

$(0.5h_w t_w f_y)/\gamma_{M0} = [(0.5\text{x}562\text{x}9.8\text{x}355)/1.0]\text{x}10^{-3} = 977.6\text{kN}$

$N_{Ed} = 140\text{kN} < 0.25N_{pl,Rd} < (0.5h_w t_w f_y)/\gamma_{M0}$

Therefore, the Bending & Axial Force Interaction checking is satisfactory.

Bending Moment Resistance:

$M_{pl,Rd} = (w_{py}f_y)/\gamma_{M0} = [(442.1\text{x}10^{-3}\text{x}355)/1.0] \text{ x}10^{-6} = 156.95\text{kNm} > M_{Ed} = 17\text{kNm}$

Therefore, the Bending Moment Resistance checking is satisfactory.

Out-of-Plane Buckling

$h/b = 597/220 = 2.71$

$t_f = 17.5\text{mm}$

curve b

$\alpha = 0.34$

$\lambda_1 = 93.9\varepsilon = 93.9\text{x}0.81 = 76.06$

$\lambda_z = \dfrac{L_{cr}}{i_z}\dfrac{1}{\lambda 1} = (1500/43.1)\text{x}(1/76.06) = 0.46$

$\emptyset_z = 0.5[1+\alpha_z(\lambda_z\text{-}0.2)\ \lambda_z^2] = 0.5[1+0.34(0.46\text{-}0.2)+0.46^2] = 0.65$

$\chi_z = \dfrac{1}{\emptyset z + \sqrt{\emptyset_z^2 - \lambda_z^2}} = \dfrac{1}{0.65 + \sqrt{0.65^2 - 0.46^2}} = 0.91$

$N_{b,z,Rd} = \chi_z A f_y/\gamma_{M1} = [(0.91\text{x}13700\text{x}355)/1.0] \text{ x}10^{-3} = 4426\text{kN} > N_{Ed} = 53\text{kN}$

Therefore, the checking is satisfactory.

Lateral Torsional Buckling Resistance, $M_{b,Rd}$:

$C_1 = 1.0$

$M_{cr} = C_1 \dfrac{\pi^2 EI_z}{L^2}\sqrt{\dfrac{I_w}{I_z} + \dfrac{L^2 GI_t}{\pi^2 EI_z}} = 1.0\dfrac{\pi^2 \times 210000 \times 3116 \times 10^4}{1500^2}\sqrt{\dfrac{1249 \times 10^9}{3116 \times 10^4} + \dfrac{1500^2 \times 81000 \times 89.3 \times 10^4}{\pi^2 \times 210000 \times 3116 \times 10^4}}$

$\qquad = 5925\text{kNm}$

$\lambda_{LT} = \sqrt{\dfrac{W_y f_y}{M_{cr}}} = \sqrt{\dfrac{442.1 \times 10^3 \times 355}{5925 \times 10^6}} = 0.16$

$\lambda_{LT,0} = 0.4$

$\beta = 0.75$

$h/b = 597/220 = 2.71$

curve c

$\lambda_{LT} = \frac{L_{cr}}{i_z}\frac{1}{\lambda 1} = (11000/43.1)\text{x}(1/76.06) = 3.36$

$\emptyset_{LT} = 0.5[1+\alpha_{LT}(\lambda_{LT}- \lambda_{LT,0})\beta\lambda_{LT}^2] = 0.5[1+0.49(0.16-0.49)+(0.75\text{x}0.16^2)] = 0.43$

$\chi_{LT} = \dfrac{1}{\emptyset LT+\sqrt{\emptyset_{lT}^2-\beta\lambda_{LT}^2}} = \dfrac{1}{0.43+\sqrt{0.43^2-0.75\times 0.16^2}} = 1.19$

$M_{b,Rd} = \dfrac{\chi_{LT}W_{pl,y}f_y}{\gamma_{M1}} = \dfrac{1.19\times 442.1\times 10^3\times 355}{1.0}$ x10^{-6} = 186.72kNm > M$_{Ed}$ = 2kNm

Therefore, the checking is satisfactory.

Interaction of Axial Force and Bending Moment:

$C_{mLT} = 1.0$

$K_{zy} = max\left[\left(1 - \dfrac{0.1\lambda_z}{C_{mLT}-0.25}\dfrac{N_{Ed}}{N_{b,Rd,z}}\right); \left(1 - \dfrac{0.1}{C_{mLT}-0.25}\dfrac{N_{Ed}}{N_{b,Rd,z}}\right)\right]$

$K_{zy} = max\left[\left(1 - \dfrac{0.1\times 0.46}{1-0.25}\dfrac{53}{4426}\right); \left(1 - \dfrac{0.1}{1-0.25}\dfrac{53}{4426}\right)\right] = max[1;1] = 1$

$\dfrac{N_{Ed}}{N_{b,z,Rd}} + k_{zy}\dfrac{M_{y,Ed}}{M_{b,Rd}} \leq 1.0$

$\dfrac{53}{4426} + 1\dfrac{2}{186.72} \leq 1.0 = 0.02 < 1.0$

Therefore, the checking is satisfactory.

<u>In Plane Buckling</u>

$h/b = 597/220 = 2.71$

$t_f = 17.5mm$

buckling about y axis

curve a

$\alpha = 0.21$

$\lambda_1 = 93.9\varepsilon = 93.9 \times 0.81 = 76.06$

$\lambda_y = \frac{L_{cr}}{i_y} \frac{1}{\lambda 1} = (6083/43.1) \times (1/76.06) = 1.86$

$\emptyset_y = 0.5[1+\alpha_y(\lambda_y\text{-}0.2)\,\lambda_y^2] = 0.5[1+0.21(1.86\text{-}0.2)+1.86^2] = 2.4$

$\chi_y = \frac{1}{\emptyset y + \sqrt{\emptyset_y^2 - \lambda_y^2}} = \frac{1}{2.4 + \sqrt{2.4^2 - 1.86^2}} = 0.26$

$N_{b,y,Rd} = \chi_y A f_y / \gamma_{M1} = [(0.26 \times 13700 \times 355)/1.0]\ \times 10^{-3} = 124.67\text{kN} > N_{Ed} = 53\text{kN}$

Therefore, the checking is satisfactory.

Lateral Torsional Buckling Resistance, $M_{b,Rd}$:

$M_{b,Rd} = 186.72\text{kNm}$

Interaction of Axial Force and Bending Moment:

$\alpha_{h\,=\,1}$

$C_{my} = 0.95 + 0.05\alpha_h = 0.95 + 0.05(1) = 1$

$K_{zy} = min\left[\left(C_{my}(1 + (\lambda_y - 0.2)\frac{N_{Ed}}{N_{b,y,Rd}})\right); \left(C_{my}(1 + 0.8\frac{N_{Ed}}{N_{b,y,Rd}})\right)\right]$

$K_{zy} = min\left[\left(1(1 + (1.86 - 0.2)\frac{53}{124.67})\right); \left(1(1 + 0.8\frac{53}{124.67})\right)\right] = min[1.7; 1.34] = 1.34$

$\frac{N_{Ed}}{N_{b,z,Rd}} + k_{zy}\frac{M_{y,Ed}}{M_{b,Rd}} \leq 1.0$

$\frac{53}{124.67} + 1.34\frac{2}{186.72} \leq 1.0 = 0.44 < 1.0$

Therefore, the checking is satisfactory.

APPENDIX A2

STIFFNESS MATRIX OF PORTAL FRAME
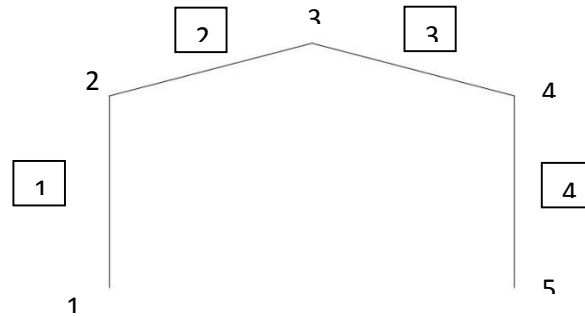
$$C = \cos\theta = \frac{x_j - x_i}{L} \qquad S = \sin\theta = \frac{y_j - y_i}{L}$$

$$
k = \frac{E}{L}
\begin{bmatrix}
AC^2 + \dfrac{12I}{L^2}S^2 & \left(A - \dfrac{12I}{L^2}\right)CS & \left(\dfrac{-6I}{L}\right)S & -\left(AC^2 + \dfrac{12I}{L^2}S^2\right) & -\left(A - \dfrac{12I}{L^2}\right)CS & \dfrac{-6I}{L}S \\[2em]
 & AS^2 + \dfrac{12I}{L^2}C^2 & \left(\dfrac{+6I}{L}\right)C & -\left(A - \dfrac{12I}{L^2}\right)CS & -\left(AS^2 + \dfrac{12I}{L^2}C^2\right) & \dfrac{6I}{L}C \\[2em]
 & & 4I & \dfrac{6I}{L}S & -\dfrac{6I}{L}C & 2I \\[2em]
 & & & AC^2 + \dfrac{12I}{L^2}S^2 & \left[A - \dfrac{12I}{L^2}\right]CS & \dfrac{6I}{L}S \\[2em]
 & \text{Symmetry} & & & AS^2 + \dfrac{12I}{L^2}C^2 & -\dfrac{6I}{L}C \\[2em]
 & & & & & 4I
\end{bmatrix}
$$

$I = 1.188\text{E}+06\text{mm}^4$

$A = 1.370\text{E}+04\text{mm}^2$

$E = 2.00\text{E}+05\text{N/mm}^2$

Element 1

$L = 11000$mm

$X_1 = 0$

$X_2 = 0$

$Y_1 = 0$

$Y_2 = 11$

$C = 0$

$S = 1$

$$k = \begin{bmatrix} 2.14 & 0 & -11781.82 & -2.14 & 0 & 11781.82 \\ 0 & 249091 & 0 & 0 & -249090.9 & 0 \\ -11782 & 0 & 8.64 \times 10^7 & 11781.82 & 0 & 4.32 \times 10^7 \\ -2.14 & 0 & 11781.82 & 2.14 & 0 & 1.18 \times 10^4 \\ 0 & -2 \times 10^5 & 0 & 0 & 249090.9 & 0 \\ -11782 & 0 & 4.32 \times 10^7 & 11781.82 & 0 & 8.64 \times 10^7 \end{bmatrix}$$

Element 2

$L = 6080$mm

$X_1 = 0$

$X_2 = 6$

$Y_1 = 0$

$Y_2 = 1$

$C = 0.9868$

$S = 0.1645$

$k =$

$$\begin{bmatrix} 438876.8 & 73144.02 & -6342.89 & -438876.8 & -73144.02 & -6342.89 \\ 73144.02 & 12203.37 & 38057.32 & -2223.58 & -12203.37 & 38057.32 \\ -6342.89 & 38057.32 & 1.563 \times 10^8 & 6342.89 & -38057.32 & 7.816 \times 10^7 \\ -438876.8 & -2223.58 & 6342.89 & 438876.8 & 73144.02 & 6342.89 \\ -7.3 \times 10^4 & 12203.37 & -38057.32 & 73144.02 & 12203.37 & -38057.32 \\ -6342.89 & 38057.32 & 7.816 \times 10^7 & 6342.89 & -38057.32 & 1.563 \times 10^8 \end{bmatrix}$$

Element 3

$L = 6080$mm

$X_1 = 0$

$X_2 = 6$

$Y_1 = 0$

$Y_2 = -1$

$C = 0.9868$

$S = -0.1645$

$k =$

$$
\begin{bmatrix}
438876.8 & -73144.02 & 6342.89 & -438876.8 & 73144.02 & 6342.89 \\
-73144.02 & 12203.37 & 38057.32 & 2223.58 & -12203.37 & 38057.32 \\
6342.89 & 38057.32 & 1.563 \times 10^8 & -6342.89 & -38057.32 & 7.816 \times 10^7 \\
-438876.8 & 2223.58 & -6342.89 & 438876.8 & -73144.02 & -6342.89 \\
7.3 \times 10^4 & -12203.37 & -38057.32 & -73144.02 & 12203.37 & -38057.32 \\
6342.89 & 38057.32 & 7.816 \times 10^7 & -6342.89 & -38057.32 & 1.563 \times 10^8
\end{bmatrix}
$$

Element 4

$L = 11000$mm $Y_2 = -11$

$X_1 = 0$

$X_2 = 0$ $C = 0$

$Y_1 = 0$ $S = -1$

$$
k =
\begin{bmatrix}
2.14 & 0 & 11781.82 & -2.14 & 0 & 11781.82 \\
0 & 249091 & 0 & 0 & -249090.9 & 0 \\
11782 & 0 & 8.64 \times 10^7 & -11781.82 & 0 & 4.32 \times 10^7 \\
-2.14 & 0 & -11781.82 & 2.14 & 0 & -1.18 \times 10^4 \\
0 & -2 \times 10^5 & 0 & 0 & 249090.9 & 0 \\
11782 & 0 & 4.32 \times 10^7 & -11781.82 & 0 & 8.64 \times 10^7
\end{bmatrix}
$$

Total Stiffness Matrix

$$K = \times 10^5 \begin{bmatrix}
0.0002 & 0 & -0.12 & 0.00002 & 0 & -0.12 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 2.5 & 0 & 0 & -2.5 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
-0.12 & 0 & 864 & 0.12 & 0 & 432 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0.0002 & 0 & 0.12 & 4.39 & 0.7 & 0.05 & -4.4 & -0.73 & -0.06 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & -2.5 & 0 & 0.73 & 2.6 & 0.38 & -0.02 & -0.12 & 0.38 & 0 & 0 & 0 & 0 & 0 & 0 \\
-0.12 & 0 & 432 & 0.05 & 0.38 & 2427 & 0.06 & -0.38 & 782 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & -4.4 & -0.02 & 0.06 & 8.8 & 0 & 0.13 & -4.4 & 0.73 & 0.06 & 0 & 0 & 0 \\
0 & 0 & 0 & -0.7 & -0.12 & -0.38 & 0 & 0.24 & 0 & 0.02 & -0.12 & 0.38 & 0 & 0 & 0 \\
0 & 0 & 0 & -0.06 & 0.38 & 782 & 0.13 & 0 & 3126 & -0.06 & -0.38 & 782 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & -4.4 & 0.02 & -0.06 & 4.4 & -0.73 & 0.05 & -0.0002 & 0 & 0.12 \\
0 & 0 & 0 & 0 & 0 & 0 & 0.73 & -0.12 & -0.38 & -0.73 & 2.61 & -0.38 & 0 & -2.5 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0.06 & 0.38 & 782 & 0.05 & -0.38 & 2427 & -0.12 & 0 & 432 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -0.0002 & 0 & -0.12 & 0.00002 & 0 & -0.12 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -2.5 & 0 & 0 & 2.5 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0.12 & 0 & 432 & -0.12 & 0 & 864
\end{bmatrix}$$

APPENDIX A3

DISPLACEMENTS OF PORTAL FRAME

$[D] = [K]^T [F]$

$$
\begin{bmatrix} u_2 \\ v_2 \\ \emptyset_2 \\ u_3 \\ v_3 \\ \emptyset_3 \\ u_4 \\ v_4 \\ \emptyset_4 \end{bmatrix}
$$

$$
= \times 10^8 \begin{bmatrix}
-463 & 62.9 & -0.19 & -456 & -1347 & -0.03 & -459.8 & -64 & 0.23 \\
62.9 & 383.6 & 0.01 & 62.9 & 383.6 & -0.03 & 64 & 18.2 & -0.05 \\
-0.19 & 0.01 & 0.45 & -0.2 & 0.09 & -0.12 & -0.2 & -0.01 & 0.04 \\
-455 & 62.9 & -0.2 & -220 & -1347 & -0.03 & -224.8 & -64 & 0.22 \\
-1347 & 383.6 & 0.09 & -1347 & 383.5 & -0.07 & -1346 & 18.21 & 0.03 \\
-0.03 & -0.03 & -0.12 & -0.03 & -0.07 & 0.38 & -0.01 & 0.04 & -0.12 \\
-459.8 & 64 & -0.2 & -225 & -1346 & -0.01 & 10.37 & 3 & 0.22 \\
-64 & 18.2 & -0.01 & -64 & 18.2 & 0.038 & 3 & 402 & 0.05 \\
0.23 & -0.05 & 0.04 & 0.22 & 0.03 & -0.12 & 0.22 & 0.05 & 0.45
\end{bmatrix}
\begin{bmatrix} 0 \\ -50000 \\ 0 \\ 0 \\ -50000 \\ 0 \\ 0 \\ -50000 \\ 0 \end{bmatrix}
$$

$$
\begin{bmatrix} u_2 \\ v_2 \\ \emptyset_2 \\ u_3 \\ v_3 \\ \emptyset_3 \\ u_4 \\ v_4 \\ \emptyset_4 \end{bmatrix}
=
\begin{bmatrix} 0.67 \\ -0.39 \\ -4.41(10^{-5}) \\ 0.67 \\ -0.39 \\ -3.55(10^{-5}) \\ 0.64 \\ -0.22 \\ -1.62(10^{-5}) \end{bmatrix}
\begin{matrix} m \\ m \\ rad \\ m \\ m \\ rad \\ m \\ m \\ rad \end{matrix}
$$

APPENDIX A4

ELEMENT FORCES OF PORTAL FRAME

$$[T]\{d\} = \begin{bmatrix} C & S & 0 & 0 & 0 & 0 \\ -S & C & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & C & S & 0 \\ 0 & 0 & 0 & -S & C & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} u_i \\ v_i \\ \phi_i \\ u_j \\ v_j \\ \phi_j \end{bmatrix}$$

$$k' = \begin{bmatrix} C_1 & 0 & 0 & -C_1 & 0 & 0 \\ 0 & 12C_2 & 6C_2L & 0 & -12C_2 & 6C_2L \\ 0 & 6C_2L & 4C_2L^2 & 0 & -6C_2L & 2C_2L^2 \\ -C_1 & 0 & 0 & C_1 & 0 & 0 \\ 0 & -12C_2 & -6C_2L & 0 & 12C_2 & -6C_2L \\ 0 & 6C_2L & 2C_2L^2 & 0 & -6C_2L & 4C_2L^2 \end{bmatrix}$$

$$\{f'\} = [k'][T]\{d\} - \{f_0\}$$

$$C_1 = \frac{AE}{L} \quad C_2 = \frac{EI}{L^3}$$

Element 1

$c = 0$

$s = 1$

$C_1 = 249090.9$

$C_2 = 0.1785$

$L = 11000mm$

$$Td= \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0.67 \\ -0.39 \\ -4.4(10^{-5}) \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ -0.39 \\ -0.67 \\ -4.4(10^{-5}) \end{bmatrix}$$

$[f'] = [k'] \, [Td]$

$$[f'] = \times 10^5 \begin{bmatrix} 2.5 & 0 & 0 & -2.5 & 0 & 0 \\ 0 & 0.00002 & 0.12 & 0 & -0.00002 & 0.12 \\ 0 & 0.12 & 864 & 0 & -0.12 & 432 \\ 2.5 & 0 & 0 & 2.5 & 1 & 0 \\ 0 & -0.00002 & -0.12 & 0 & 0.00002 & -0.12 \\ 0 & 0.12 & 432 & 0 & -0.12 & 864 \end{bmatrix} \begin{bmatrix} 0 \\ 0 \\ 0 \\ -0.39 \\ -0.67 \\ -4.4(10^{-5}) \end{bmatrix}$$

$$= \begin{bmatrix} 97809.4 \\ 0.92 \\ 6035.5 \\ -97809.4 \\ -0.924 \\ 4128.45 \end{bmatrix} \begin{matrix} N \\ N \\ N.m \\ N \\ N \\ N.m \end{matrix}$$

Element 2

$c = 0.9868$

$s = 0.1645$

$C_1 = 450657.9$

$C_2 = 1.0571$

$L = 6080mm$

$$Td= \begin{bmatrix} 0.9868 & 0.1645 & 0 & 0 & 0 & 0 \\ -0.1645 & 0.9868 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.9868 & 0.1645 & 0 \\ 0 & 0 & 0 & -0.1645 & 0.9868 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} 0.67 \\ -0.39 \\ -4.4(10^{-5}) \\ 0.67 \\ -0.39 \\ -3.6(10^{-5}) \end{bmatrix}$$

$$= \begin{bmatrix} 0.6 \\ -0.5 \\ -4.4(10^{-5}) \\ 0.6 \\ -0.49 \\ -3.6(10^{-5}) \end{bmatrix}$$

$$[f'] = [k'] [Td]$$

$$[f'] = \times 10^5 \begin{bmatrix} 4.5 & 0 & 0 & -4.5 & 0 & 0 \\ 0 & 0.00013 & 0.39 & 0 & -0.00013 & 0.38 \\ 0 & 0.39 & 1560 & 0 & -0.39 & 782 \\ 4.5 & 0 & 0 & 4.5 & 0 & 0 \\ 0 & -0.00013 & -0.38 & 0 & 0.00013 & -0.38 \\ 0 & 0.39 & 782 & 0 & -0.38 & 1560 \end{bmatrix} \begin{bmatrix} 0.6 \\ -0.5 \\ -4.4(10^{-5}) \\ 0.6 \\ -0.49 \\ -3.6(10^{-5}) \end{bmatrix}$$

$$= \begin{bmatrix} -0.99 \\ -0.33 \\ -4128.45 \\ 541406.6 \\ 0.3344 \\ 2095.04 \end{bmatrix} \begin{matrix} N \\ N \\ N.m \\ N \\ N \\ N.m \end{matrix}$$

Element 3

$c = 0.9868$

$s = -0.1645$

$C_1 = 450657.9$

$C_2 = 1.0571$

$L = 6080mm$

$$Td = \begin{bmatrix} 0.9868 & -0.1645 & 0 & 0 & 0 & 0 \\ 0.1645 & 0.9868 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.9868 & -0.1645 & 0 \\ 0 & 0 & 0 & 0.1645 & 0.9868 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} 0.67 \\ -0.39 \\ -3.6(10^{-5}) \\ 0.64 \\ -0.22 \\ -1.6(10^{-5}) \end{bmatrix}$$

$$= \begin{bmatrix} 0.73 \\ -0.28 \\ 3.55(10^{-5}) \\ 0.67 \\ -0.11 \\ -1.6(10^{-5}) \end{bmatrix}$$

[f'] = [k'] [Td]

$$[f'] = \times 10^5 \begin{bmatrix} 4.5 & 0 & 0 & -4.5 & 0 & 0 \\ 0 & 0.00013 & 0.39 & 0 & -0.00013 & 0.38 \\ 0 & 0.39 & 1560 & 0 & -0.39 & 782 \\ 4.5 & 0 & 0 & 4.5 & 0 & 0 \\ 0 & -0.00013 & -0.38 & 0 & 0.00013 & -0.38 \\ 0 & 0.39 & 782 & 0 & -0.38 & 1560 \end{bmatrix} \begin{bmatrix} 0.73 \\ -0.28 \\ 3.55(10^{-5}) \\ 0.67 \\ -0.11 \\ -1.6(10^{-5}) \end{bmatrix}$$

$$= \begin{bmatrix} 28218.61 \\ -1.35 \\ -2095.04 \\ 629599.8 \\ 1.35 \\ -6134.94 \end{bmatrix} \begin{matrix} N \\ N \\ N.m \\ N \\ N \\ N.m \end{matrix}$$

Element 4

$c = 0$

$s = -1$

$C_1 = 249090.9$

$C_2 = 0.1785$

$L = 11000mm$

$$Td = \begin{bmatrix} 0 & -1 & 0 & 0 & 0 & 0 \\ 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} 0.64 \\ -0.22 \\ -1.6(10^{-5}) \\ 0 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} 0.22 \\ 0.64 \\ -1.6(10^{-5}) \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

[f'] = [k'] [Td]

$$[f'] = \times 10^5 \begin{bmatrix} 2.5 & 0 & 0 & -2.5 & 0 & 0 \\ 0 & 0.00002 & 0.12 & 0 & -0.00002 & 0.12 \\ 0 & 0.12 & 864 & 0 & -0.12 & 432 \\ 2.5 & 0 & 0 & 2.5 & 1 & 0 \\ 0 & -0.00002 & -0.12 & 0 & 0.00002 & -0.12 \\ 0 & 0.12 & 432 & 0 & -0.12 & 864 \end{bmatrix} \begin{bmatrix} 0.22 \\ 0.64 \\ -1.6(10^{-5}) \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

$$= \begin{bmatrix} 54642.55 \\ 1.18 \\ 6134.94 \\ 54642.55 \\ -1.18 \\ 6835.06 \end{bmatrix} \begin{matrix} N \\ N \\ N.m \\ N \\ N \\ N.m \end{matrix}$$

APPENDIX B1

LOG FILE

~CFACTIV,CIVILFEM,Y

/TITLE,portal frame

FINISH

~UNITS,SI

/PREP7

ET,1,Beam188

~CFMP,1,LIB,STEEL,EC3, Fe E355

~SSECLIB,1,1,2,18   !IPE A 600

TEMP=20

ELASTIC=2.0

POISSON=0.2

!*

MPTEMP,,,,,,,,

MPTEMP,1,TEMP

MPDE,EX,1

MPDE,NUXY,1

MPDATA,EX,1,,ELASTICE +011

MPDATA,NUXY,1,,POISS ON

~MEMBPRO,1,EC3,ALL,3. 000,1.000,1.000,1.000,0.000, 1.000,1.000,1.000,1.000,,1,1. 000,1.000,1

~MEMBPRO,1,NLMOD,A LL,0

~MEMBPRO,1,NAME,Hori zontal Member 1

~MEMBPRO,2,EC3,ALL,2. 750,1.000,1.000,1.000,0.000, 1.000,1.000,1.000,1.000,,1,1. 000,1.000,1

~MEMBPRO,2,NLMOD,A LL,0

~MEMBPRO,2,NAME,Vert ical Member 1

~MEMBPRO,3,EC3,ALL,1. 520,1.000,1.000,1.000,0.000, 1.000,1.000,1.000,1.000,,1,1. 000,1.000,1

~MEMBPRO,3,NLMOD,A LL,0

~MEMBPRO,3,NAME,Diag onal Member 1

~MEMBPRO,4,EC3,ALL,11 .50,1.000,1.000,1.000,0.000, 1.000,1.000,1.000,1.000,,1,1. 000,1.000,1

~MEMBPRO,4,NLMOD,A LL,0

~MEMBPRO,4,NAME,Vert ical Member 2

~MEMBPRO,5,EC3,ALL,12 .00,1.000,1.000,1.000,0.000, 1.000,1.000,1.000,1.000,,1,1. 000,1.000,1

~MEMBPRO,5,NLMOD,A LL,0

~MEMBPRO,5,NAME,Vert ical Member 3

~MEMBPRO,6,EC3,ALL,40 .00,1.000,1.000,1.000,0.000, 1.000,1.000,1.000,1.000,,1,1. 000,1.000,1

~MEMBPRO,6,NLMOD,A LL,0

~MEMBPRO,6,NAME,Hori zontal Member 2

~MEMBPRO,7,EC3,ALL,5. 000,1.000,1.000,1.000,0.000, 1.000,1.000,1.000,1.000,,1,1. 000,1.000,1

~MEMBPRO,7,NLMOD,A LL,0

~MEMBPRO,7,NAME,Hori zontal Member 3

~MEMBPRO,8,EC3,ALL,10 .00,1.000,1.000,1.000,0.000, 1.000,1.000,1.000,1.000,,1,1. 000,1.000,1

~MEMBPRO,8,NLMOD,A LL,0

~MEMBPRO,8,NAME,Hori zontal Member 4

~MEMBPRO,9,EC3,ALL,7. 870,1.000,1.000,1.000,0.000, 1.000,1.000,1.000,1.000,,1,1. 000,1.000,1

~MEMBPRO,9,NLMOD,A LL,0

~MEMBPRO,9,NAME,Diag onal Member 2

~MEMBPRO,10,EC3,ALL,2 0.19,1.000,1.000,1.000,0.000 ,1.000,1.000,1.000,1.000,,1,1 .000,1.000,1

~MEMBPRO,10,NLMOD,A LL,0

~MEMBPRO,10,NAME,Dia gonal Member 5

~MEMBPRO,11,EC3,ALL,1 1.40,1.000,1.000,1.000,0.000 ,1.000,1.000,1.000,1.000,,1,1 .000,1.000,1

~MEMBPRO,11,NLMOD,A LL,0

~MEMBPRO,11,NAME,Dia gonal Member 3

~MEMBPRO,12,EC3,ALL,1 2.37,1.000,1.000,1.000,0.000 ,1.000,1.000,1.000,1.000,,1,1 .000,1.000,1

~MEMBPRO,12,NLMOD,A LL,0

~MEMBPRO,12,NAME,Dia gonal Member 4

~MEMBPRO,13,EC3,ALL,1 2.00,1.000,1.000,1.000,0.000

,1.000,1.000,1.000,1.000,,1,1
.000,1.000,1

~MEMBPRO,13,NLMOD,A
LL,0

~MEMBPRO,13,NAME,Ho
rizontal Member 5

~MEMBPRO,14,EC3,ALL,1
1.00,1.000,1.000,1.000,0.000
,1.000,1.000,1.000,1.000,,1,1
.000,1.000,1

~MEMBPRO,14,NLMOD,A
LL,0

~MEMBPRO,14,NAME,Ver
tical Member 4

~MEMBPRO,15,EC3,ALL,1
2.08,1.000,1.000,1.000,0.000
,1.000,1.000,1.000,1.000,,1,1
.000,1.000,1

~MEMBPRO,15,NLMOD,A
LL,0

~MEMBPRO,15,NAME,Dia
gonal Member 6

~MEMBPRO,16,EC3,ALL,2
0.00,1.000,1.000,1.000,0.000
,1.000,1.000,1.000,1.000,,1,1
.000,1.000,1

~MEMBPRO,16,NLMOD,A
LL,0

~MEMBPRO,16,NAME,Ho
rizontal Member 6

~MEMBPRO,17,EC3,ALL,6
.080,1.000,1.000,1.000,0.000
,1.000,1.000,1.000,1.000,,1,1
.000,1.000,1

~MEMBPRO,17,NLMOD,A
LL,0

~MEMBPRO,17,NAME,Dia
gonal Member 7

~BMSHPRO,1,BEAM,1,1,,,
188,1,0,,Horizontal Member
1

~BMSHPRO,2,BEAM,1,1,,,
188,3,0,,Vertical Member 1

~BMSHPRO,3,BEAM,1,1,,,
188,3,0,,Diagonal Member 1

~BMSHPRO,4,BEAM,1,1,,,
188,4,0,,Vertical Member 2

~BMSHPRO,5,BEAM,1,1,,,
188,5,0,,Vertical Member 3

~BMSHPRO,6,BEAM,1,1,,,
188,6,0,,Horizontal Member
2

~BMSHPRO,7,BEAM,1,1,,,
188,7,0,,Horizontal Member
3

~BMSHPRO,8,BEAM,1,1,,,
188,8,0,,Horizontal Member
4

~BMSHPRO,9,BEAM,1,1,,,
188,9,0,,Diagonal Member 2

~BMSHPRO,10,BEAM,1,1,,,
188,10,0,,Diagonal Member
5

~BMSHPRO,11,BEAM,1,1,,,
188,11,0,,Diagonal Member
3

~BMSHPRO,12,BEAM,1,1,,,
188,12,0,,Diagonal Member
4

~BMSHPRO,13,BEAM,1,1,,,
188,13,0,,Horizontal
Member 5

~BMSHPRO,14,BEAM,1,1,,,
188,14,0,,Vertical Member 4

~BMSHPRO,15,BEAM,1,1,,,
188,15,0,,Diagonal Member
6

~BMSHPRO,16,BEAM,1,1,,,
188,16,0,,Horizontal
Member 6

~BMSHPRO,17,BEAM,1,1,,,
188,17,0,,Diagonal Member
7

N,1,0,0,0,,,,

N,2,0,0,10,,,,

N,3,0,0,20,,,,

N,4,0,0,30,,,,

N,5,12,0,0,,,,

N,6,12,0,10,,,,

N,7,12,0,20,,,,

N,8,12,0,30,,,,

N,9,0,11,0,,,,

N,10,0,11,10,,,,

N,11,0,11,20,,,,

N,12,0,11,30,,,,

N,13,6,12,0,,,,

N,14,6,12,10,,,,

N,15,6,12,20,,,,

N,16,6,12,30,,,,

N,17,12,11,0,,,,

N,18,12,11,10,,,,

N,19,12,11,20,,,,

N,20,12,11,30,,,,

N,21,0,11,40,,,,

N,22,6,12,40,,,,

N,23,12,11,40,,,,

N,24,0,0,40,,,,

N,25,12,0,40,,,,

N,26,7.5,11.75,40,,,,

N,27,7.5,11.75,0,,,,

N,28,9,11.5,0,,,,

N,29,9,11.5,40,,,,

N,30,10.5,11.25,40,,,,

N,31,10.5,11.25,0,,,,

N,32,1.5,11.25,0,,,,

N,33,1.5,11.25,40,,,,

N,34,3,11.5,0,,,,

N,35,3,11.5,40,,,,

N,36,4.5,11.75,0,,,,

N,37,4.5,11.75,40,,,,

N,38,12,2.75,40,,,,

N,39,12,2.75,0,,,,

N,40,12,5.5,40,,,,

N,41,12,5.5,0,,,,

N,42,12,8.25,0,,,,

N,43,12,8.25,40,,,,

N,44,0,2.75,0,,,,

```
N,45,0,2.75,10,,,,          N,80,12,0,25,,,,          FLST,2,2,1
N,46,0,5.5,10,,,,           N,81,12,0,35,,,,          FITEM,2,38
N,47,0,5.5,0,,,,            /VIEW,1,1,2,3             FITEM,2,40
N,48,0,8.25,0,,,,           /ANG,1                    E,P51X
N,49,0,8.25,10,,,,          /REP,FAST                 FLST,2,2,1
N,50,0,2.75,30,,,,          /DIST, 1 ,0.729000,1      FITEM,2,40
N,51,0,5.5,40,,,,           /PNUM,KP,0                FITEM,2,43
N,52,0,5.5,30,,,,           /PNUM,LINE,0              E,P51X
N,53,0,8.25,30,,,,          /PNUM,AREA,0              FLST,2,2,1
N,54,0,8.25,40,,,,          /PNUM,VOLU,0              FITEM,2,43
N,55,0,2.75,40,,,,          /PNUM,NODE,1              FITEM,2,23
N,56,3,0,0,,,,              /PNUM,TABN,0              E,P51X
N,57,9,0,0,,,,              /PNUM,SVAL,0              FLST,2,2,1
N,58,3,0,40,,,,             /NUMBER,0                 FITEM,2,24
N,59,9,0,40,,,,             !*                        FITEM,2,55
N,60,6,0,0,,,,              /PNUM,ELEM,0              E,P51X
N,61,6,0,40,,,,             /REPLOT                   FLST,2,2,1
N,62,0,0,5,,,,              !*                        FITEM,2,55
N,63,0,0,15,,,,             FINISH                    FITEM,2,51
N,64,0,0,25,,,,             /SOL                      E,P51X
N,65,0,0,35,,,,             FINISH                    FLST,2,2,1
N,66,0,11,5,,,,             /POST1                    FITEM,2,51
N,67,0,11,15,,,,            FINISH                    FITEM,2,54
N,68,0,11,25,,,,            /PREP7                    E,P51X
N,69,0,11,35,,,,            TYPE,  1                  FLST,2,2,1
N,70,6,12,5,,,,             MAT,     1                FITEM,2,54
N,71,6,12,15,,,,            REAL,                     FITEM,2,21
N,72,6,12,25,,,,            ESYS,    0                E,P51X
N,73,6,12,35,,,,            SECNUM, 2                 TYPE,  1
N,74,12,11,5,,,,            TSHAP,LINE                MAT,     1
N,75,12,11,15,,,,           !*                        REAL,
N,76,12,11,25,,,,           FLST,2,2,1                ESYS,    0
N,77,12,11,35,,,,           FITEM,2,25                SECNUM,  1
N,78,12,0,5,,,,             FITEM,2,38                TSHAP,LINE
N,79,12,0,15,,,,            E,P51X                    !*
```

```
FLST,2,2,1
FITEM,2,25
FITEM,2,59
E,P51X
FLST,2,2,1
FITEM,2,59
FITEM,2,61
E,P51X
FLST,2,2,1
FITEM,2,61
FITEM,2,58
E,P51X
FLST,2,2,1
FITEM,2,58
FITEM,2,24
E,P51X
TYPE,  1
MAT,     1
REAL,
ESYS,    0
SECNUM,  3
TSHAP,LINE
!*
FLST,2,2,1
FITEM,2,23
FITEM,2,30
E,P51X
FLST,2,2,1
FITEM,2,30
FITEM,2,29
E,P51X
FLST,2,2,1
FITEM,2,29
FITEM,2,26
E,P51X

FLST,2,2,1
FITEM,2,26
FITEM,2,22
E,P51X
FLST,2,2,1
FITEM,2,22
FITEM,2,37
E,P51X
FLST,2,2,1
FITEM,2,37
FITEM,2,35
E,P51X
FLST,2,2,1
FITEM,2,35
FITEM,2,33
E,P51X
FLST,2,2,1
FITEM,2,33
FITEM,2,21
E,P51X
TYPE,  1
MAT,     1
REAL,
ESYS,    0
SECNUM,  7
TSHAP,LINE
!*
FLST,2,2,1
FITEM,2,25
FITEM,2,81
E,P51X
FLST,2,2,1
FITEM,2,81
FITEM,2,8
E,P51X

FLST,2,2,1
FITEM,2,8
FITEM,2,80
E,P51X
FLST,2,2,1
FITEM,2,80
FITEM,2,7
E,P51X
FLST,2,2,1
FITEM,2,7
FITEM,2,79
E,P51X
FLST,2,2,1
FITEM,2,6
FITEM,2,78
E,P51X
FLST,2,2,1
FITEM,2,79
FITEM,2,6
E,P51X
FLST,2,2,1
FITEM,2,78
FITEM,2,5
E,P51X
FLST,2,2,1
FITEM,2,21
FITEM,2,69
E,P51X
FLST,2,2,1
FITEM,2,69
FITEM,2,12
E,P51X
FLST,2,2,1
FITEM,2,12
FITEM,2,68
```

```
E,P51X                FITEM,2,19            TSHAP,LINE
FLST,2,2,1            E,P51X                !*
FITEM,2,68           FLST,2,2,1            FLST,2,2,1
FITEM,2,11           FITEM,2,19           FITEM,2,5
E,P51X                FITEM,2,75           FITEM,2,39
FLST,2,2,1            E,P51X                E,P51X
FITEM,2,11           FLST,2,2,1            FLST,2,2,1
FITEM,2,67           FITEM,2,75           FITEM,2,39
E,P51X                FITEM,2,18           FITEM,2,41
FLST,2,2,1            E,P51X                E,P51X
FITEM,2,67           FLST,2,2,1            FLST,2,2,1
FITEM,2,10           FITEM,2,18           FITEM,2,41
E,P51X                FITEM,2,74           FITEM,2,42
FLST,2,2,1            E,P51X                E,P51X
FITEM,2,10           FLST,2,2,1            FLST,2,2,1
FITEM,2,66           FITEM,2,74           FITEM,2,42
E,P51X                FITEM,2,17           FITEM,2,17
FLST,2,2,1            E,P51X                E,P51X
FITEM,2,66           FLST,2,2,1            FLST,2,2,1
FITEM,2,9            FITEM,2,13           FITEM,2,1
E,P51X                FITEM,2,70           FITEM,2,44
FLST,2,2,1            E,P51X                E,P51X
FITEM,2,23           FLST,2,2,1            FLST,2,2,1
FITEM,2,77           FITEM,2,70           FITEM,2,44
E,P51X                FITEM,2,14           FITEM,2,47
FLST,2,2,1            E,P51X                E,P51X
FITEM,2,77           FLST,2,2,1            FLST,2,2,1
FITEM,2,20           FITEM,2,14           FITEM,2,47
E,P51X                FITEM,2,71           FITEM,2,48
FLST,2,2,1            E,P51X                E,P51X
FITEM,2,20           TYPE,  1             FLST,2,2,1
FITEM,2,76           MAT,    1            FITEM,2,48
E,P51X                REAL,                FITEM,2,9
FLST,2,2,1            ESYS,    0           E,P51X
FITEM,2,76           SECNUM,  2           TYPE,  1
```

```
MAT,    1
REAL,
ESYS,    0
SECNUM,  3
TSHAP,LINE
!*
FLST,2,2,1
FITEM,2,17
FITEM,2,31
E,P51X
FLST,2,2,1
FITEM,2,31
FITEM,2,28
E,P51X
FLST,2,2,1
FITEM,2,28
FITEM,2,27
E,P51X
FLST,2,2,1
FITEM,2,27
FITEM,2,13
E,P51X
FLST,2,2,1
FITEM,2,13
FITEM,2,36
E,P51X
FLST,2,2,1
FITEM,2,36
FITEM,2,34
E,P51X
FLST,2,2,1
FITEM,2,34
FITEM,2,32
E,P51X
FLST,2,2,1
```

```
FITEM,2,32
FITEM,2,9
E,P51X
GPLOT
TYPE,  1
MAT,    1
REAL,
ESYS,    0
SECNUM,  13
TSHAP,LINE
!*
FLST,2,2,1
FITEM,2,54
FITEM,2,43
E,P51X
FLST,2,2,1
FITEM,2,51
FITEM,2,40
E,P51X
FLST,2,2,1
FITEM,2,55
FITEM,2,38
E,P51X
FLST,2,2,1
FITEM,2,48
FITEM,2,42
E,P51X
FLST,2,2,1
FITEM,2,47
FITEM,2,41
E,P51X
FLST,2,2,1
FITEM,2,44
FITEM,2,39
E,P51X
```

```
TYPE,  1
MAT,    1
REAL,
ESYS,    0
SECNUM,  1
TSHAP,LINE
!*
FLST,2,2,1
FITEM,2,5
FITEM,2,57
E,P51X
FLST,2,2,1
FITEM,2,57
FITEM,2,60
E,P51X
FLST,2,2,1
FITEM,2,60
FITEM,2,56
E,P51X
FLST,2,2,1
FITEM,2,56
FITEM,2,1
E,P51X
TYPE,  1
MAT,    1
REAL,
ESYS,    0
SECNUM,  6
TSHAP,LINE
!*
FLST,2,2,1
FITEM,2,43
FITEM,2,42
E,P51X
FLST,2,2,1
```

```
FITEM,2,40          FITEM,2,18          TSHAP,LINE
FITEM,2,41          FITEM,2,6           !*
E,P51X              E,P51X              FLST,2,2,1
FLST,2,2,1          FLST,2,2,1          FITEM,2,22
FITEM,2,38          FITEM,2,74          FITEM,2,61
FITEM,2,39          FITEM,2,78          E,P51X
E,P51X              E,P51X              FLST,2,2,1
TYPE,  1            TYPE,  1            FITEM,2,13
MAT,    1           MAT,    1           FITEM,2,60
REAL,               REAL,               E,P51X
ESYS,   0           ESYS,   0           TYPE,  1
SECNUM, 14          SECNUM, 4           MAT,    1
TSHAP,LINE          TSHAP,LINE          REAL,
!*                  !*                  ESYS,   0
FLST,2,2,1          FLST,2,2,1          SECNUM, 11
FITEM,2,77          FITEM,2,28          TSHAP,LINE
FITEM,2,81          FITEM,2,57          !*
E,P51X              E,P51X              FLST,2,2,1
FLST,2,2,1          FLST,2,2,1          FITEM,2,23
FITEM,2,20          FITEM,2,34          FITEM,2,59
FITEM,2,8           FITEM,2,56          E,P51X
E,P51X              E,P51X              FLST,2,2,1
FLST,2,2,1          FLST,2,2,1          FITEM,2,21
FITEM,2,76          FITEM,2,29          FITEM,2,58
FITEM,2,80          FITEM,2,59          E,P51X
E,P51X              E,P51X              FLST,2,2,1
FLST,2,2,1          FLST,2,2,1          FITEM,2,17
FITEM,2,19          FITEM,2,35          FITEM,2,57
FITEM,2,7           FITEM,2,58          E,P51X
E,P51X              E,P51X              FLST,2,2,1
FLST,2,2,1          TYPE,  1            FITEM,2,9
FITEM,2,75          MAT,    1           FITEM,2,56
FITEM,2,79          REAL,               E,P51X
E,P51X              ESYS,   0           TYPE,  1
FLST,2,2,1          SECNUM, 5           MAT,    1
```

```
REAL,                    E,P51X                   E,P51X
ESYS,    0               TYPE,  1                  TYPE,  1
SECNUM,  12              MAT,     1                MAT,     1
TSHAP,LINE               REAL,                    REAL,
!*                       ESYS,    0               ESYS,    0
FLST,2,2,1               SECNUM,  15              SECNUM,  7
FITEM,2,34               TSHAP,LINE               TSHAP,LINE
FITEM,2,1                !*                       !*
E,P51X                   FLST,2,2,1               FLST,2,2,1
FLST,2,2,1               FITEM,2,77               FITEM,2,71
FITEM,2,28               FITEM,2,25               FITEM,2,15
FITEM,2,5                E,P51X                   E,P51X
E,P51X                   FLST,2,2,1               FLST,2,2,1
FLST,2,2,1               FITEM,2,23               FITEM,2,15
FITEM,2,29               FITEM,2,81               FITEM,2,72
FITEM,2,25               E,P51X                   E,P51X
E,P51X                   FLST,2,2,1               FLST,2,2,1
FLST,2,2,1               FITEM,2,17               FITEM,2,72
FITEM,2,35               FITEM,2,78               FITEM,2,16
FITEM,2,24               E,P51X                   E,P51X
E,P51X                   FLST,2,2,1               FLST,2,2,1
TYPE,  1                 FITEM,2,74               FITEM,2,16
MAT,     1               FITEM,2,5                FITEM,2,73
REAL,                    E,P51X                   E,P51X
ESYS,    0               FLST,2,2,1               FLST,2,2,1
SECNUM,  13              FITEM,2,65               FITEM,2,73
TSHAP,LINE               FITEM,2,21               FITEM,2,22
!*                       E,P51X                   E,P51X
FLST,2,2,1               FLST,2,2,1               FLST,2,2,1
FITEM,2,17               FITEM,2,69               FITEM,2,24
FITEM,2,9                FITEM,2,24               FITEM,2,65
E,P51X                   E,P51X                   E,P51X
FLST,2,2,1               FLST,2,2,1               FLST,2,2,1
FITEM,2,23               FITEM,2,66               FITEM,2,65
FITEM,2,21               FITEM,2,1                FITEM,2,4
```

```
E,P51X                E,P51X                !*
FLST,2,2,1            FLST,2,2,1            FLST,2,2,1
FITEM,2,4            FITEM,2,50           FITEM,2,69
FITEM,2,64           FITEM,2,52           FITEM,2,65
E,P51X                E,P51X                E,P51X
FLST,2,2,1            FLST,2,2,1            FLST,2,2,1
FITEM,2,64           FITEM,2,52           FITEM,2,62
FITEM,2,3            FITEM,2,53           FITEM,2,66
E,P51X                E,P51X                E,P51X
FLST,2,2,1            FLST,2,2,1            TYPE,  1
FITEM,2,3            FITEM,2,53           MAT,    1
FITEM,2,63           FITEM,2,12           REAL,
E,P51X                E,P51X                ESYS,    0
FLST,2,2,1            FLST,2,2,1            SECNUM,  8
FITEM,2,63           FITEM,2,2            TSHAP,LINE
FITEM,2,2            FITEM,2,45           !*
E,P51X                E,P51X                FLST,2,2,1
FLST,2,2,1            FLST,2,2,1            FITEM,2,54
FITEM,2,2            FITEM,2,45           FITEM,2,53
FITEM,2,62           FITEM,2,46           E,P51X
E,P51X                E,P51X                FLST,2,2,1
FLST,2,2,1            FLST,2,2,1            FITEM,2,51
FITEM,2,62           FITEM,2,46           FITEM,2,52
FITEM,2,1            FITEM,2,49           E,P51X
E,P51X                E,P51X                FLST,2,2,1
TYPE,  1             FLST,2,2,1            FITEM,2,55
MAT,    1            FITEM,2,49           FITEM,2,50
REAL,                FITEM,2,10           E,P51X
ESYS,    0            E,P51X                FLST,2,2,1
SECNUM,  2           TYPE,  1             FITEM,2,45
TSHAP,LINE           MAT,    1            FITEM,2,44
!*                    REAL,                E,P51X
FLST,2,2,1            ESYS,    0            FLST,2,2,1
FITEM,2,4            SECNUM, 14           FITEM,2,46
FITEM,2,50           TSHAP,LINE           FITEM,2,47
```

```
E,P51X              FLST,2,2,1          FLST,2,2,1
FLST,2,2,1          FITEM,2,46          FITEM,2,11
FITEM,2,49          FITEM,2,52          FITEM,2,15
FITEM,2,48          E,P51X              E,P51X
E,P51X              TYPE,  1            FLST,2,2,1
TYPE,  1            MAT,     1          FITEM,2,15
MAT,     1          REAL,               FITEM,2,19
REAL,               ESYS,    0          E,P51X
ESYS,    0          SECNUM, 17          FLST,2,2,1
SECNUM, 10          TSHAP,LINE          FITEM,2,68
TSHAP,LINE          !*                  FITEM,2,72
!*                  FLST,2,2,1          E,P51X
FLST,2,2,1          FITEM,2,66          FLST,2,2,1
FITEM,2,10          FITEM,2,70          FITEM,2,72
FITEM,2,53          E,P51X              FITEM,2,76
E,P51X              FLST,2,2,1          E,P51X
FLST,2,2,1          FITEM,2,70          FLST,2,2,1
FITEM,2,12          FITEM,2,74          FITEM,2,12
FITEM,2,49          E,P51X              FITEM,2,16
E,P51X              FLST,2,2,1          E,P51X
FLST,2,2,1          FITEM,2,10          FLST,2,2,1
FITEM,2,46          FITEM,2,14          FITEM,2,16
FITEM,2,53          E,P51X              FITEM,2,20
E,P51X              FLST,2,2,1          E,P51X
FLST,2,2,1          FITEM,2,14          FLST,2,2,1
FITEM,2,49          FITEM,2,18          FITEM,2,77
FITEM,2,52          E,P51X              FITEM,2,73
E,P51X              FLST,2,2,1          E,P51X
TYPE,  1            FITEM,2,75          FLST,2,2,1
MAT,     1          FITEM,2,71          FITEM,2,73
REAL,               E,P51X              FITEM,2,69
ESYS,    0          FLST,2,2,1          E,P51X
SECNUM, 16          FITEM,2,71          TYPE,  1
TSHAP,LINE          FITEM,2,67          MAT,     1
!*                  E,P51X              REAL,
```

```
ESYS,    0              FITEM,2,20             FITEM,2,73
SECNUM, 16              E,P51X                 E,P51X
TSHAP,LINE             FLST,2,2,1             FLST,2,2,1
!*                      FITEM,2,20             FITEM,2,73
FLST,2,2,1             FITEM,2,72             FITEM,2,12
FITEM,2,53             E,P51X                 E,P51X
FITEM,2,49             FLST,2,2,1             FLST,2,2,1
E,P51X                 FITEM,2,72             FITEM,2,12
TYPE, 1                FITEM,2,19             FITEM,2,72
MAT,     1             E,P51X                 E,P51X
REAL,                  FLST,2,2,1             FLST,2,2,1
ESYS,    0             FITEM,2,19             FITEM,2,72
SECNUM, 15             FITEM,2,71             FITEM,2,11
TSHAP,LINE            E,P51X                 E,P51X
!*                     FLST,2,2,1             FLST,2,2,1
FLST,2,2,1            FITEM,2,71             FITEM,2,11
FITEM,2,9            FITEM,2,18             FITEM,2,71
FITEM,2,62           E,P51X                 E,P51X
E,P51X               FLST,2,2,1             FLST,2,2,1
TYPE, 1              FITEM,2,18             FITEM,2,71
MAT,     1           FITEM,2,70             FITEM,2,10
REAL,                E,P51X                 E,P51X
ESYS,    0           FLST,2,2,1             FLST,2,2,1
SECNUM,  9           FITEM,2,70             FITEM,2,10
TSHAP,LINE          FITEM,2,17             FITEM,2,70
!*                   E,P51X                 E,P51X
/VIEW,1,1,2,3        TYPE, 1                FLST,2,2,1
/ANG,1              MAT,     1             FITEM,2,70
/REP,FAST           REAL,                  FITEM,2,9
FLST,2,2,1          ESYS,    0             E,P51X
FITEM,2,23          SECNUM,  9             FLST,2,2,1
FITEM,2,73          TSHAP,LINE            FITEM,2,16
E,P51X              !*                     FITEM,2,68
FLST,2,2,1          FLST,2,2,1             E,P51X
FITEM,2,73          FITEM,2,21             FLST,2,2,1
```

```
FITEM,2,68
FITEM,2,15
E,P51X
FLST,2,2,1
FITEM,2,15
FITEM,2,67
E,P51X
FLST,2,2,1
FITEM,2,67
FITEM,2,14
E,P51X
FINISH
/SOL
FLST,2,7,1,ORDE,5
FITEM,2,5
FITEM,2,-8
FITEM,2,25
FITEM,2,78
FITEM,2,81
!*
/GO
D,P51X, ,0, , , ,ALL, , , , ,
FLST,2,7,1,ORDE,5
FITEM,2,1
FITEM,2,-4
FITEM,2,24
FITEM,2,62
FITEM,2,65
!*
/GO
D,P51X, ,0, , , ,ALL, , , , ,
FLST,2,21,1,ORDE,8
FITEM,2,9
FITEM,2,-23
FITEM,2,66
```

```
FITEM,2,69
FITEM,2,-70
FITEM,2,73
FITEM,2,-74
FITEM,2,77
!*
/GO
LOAD=-50000
WINDLOAD=15000
F,P51X,FY,LOAD
FLST,2,20,2,ORDE,2
FITEM,2,1
FITEM,2,-20
FLST,2,20,2,ORDE,2
FITEM,2,1
FITEM,2,-20
SFBEAM,P51X,1,PRES,0.8
*WINDLOAD, , , , , ,0
/STATUS,SOLU
SOLVE
FINISH
/POST1
PLDISP,2
!*
/EFACET,1
PLNSOL, U,SUM, 0,1.0
~CHKSTL,TENSION,-
Z,PARTIAL
~CFSET,0,1,LAST,
~CHKSTL,TENSION,-
Z,PARTIAL
~PLLSSTL,ELM_OK,1,
! Sort the displacements at
nodes in ascending
! order according to absolute
values
nsort,U,Y,1,1
```

```
! Dig out the maximum
absolute displacement


*GET,MAX_Deflection,sort,
0,max

nsel,all

NSORT,U,SUM,1,1

*GET,Maximum
Deflection,SORT,0,MAX

nsel,all

NSORT,EPTO,EQV

*GET,Max   Von   Mises
Strain,SORT,0,MAX

nsel,all

NSORT,s,EQV

*GET,Max   Von   Mises
Stress,SORT,0,MAX
```