ANALYSIS OF STEEL STRUCTURE
FOR 5-STORIES
APARTMENT USING ANSYS

OOI YOONG KAM

B. ENG(HONS.) CIVIL ENGINEERING

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

DECLARATION OF THESIS AND COPYRIGHT

Author's Full Name : OOI YOONG KAM

Date of Birth : 29TH DECEMBER 1995

Title : ANALYSIS OF STEEL STRUCTURE FOR 5-STORIES APARTMENT USING ANSYS

Academic Session : SEMESTER II 2018/2019

I declare that this thesis is classified as:

☐	CONFIDENTIAL	(Contains confidential information under the Official Secret Act 1997)*
☐	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 reserves the following rights:

1. The Thesis is the Property of Universiti Malaysia Pahang
2. The Library of Universiti Malaysia Pahang has the right to make copies of the thesis 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)	(Supervisor's Signature)
951229-08-5051	DR. CHENG HOCK TIAN
New IC/Passport Number	Name of Supervisor
Date: 31 MAY 2019	Date: 31 MAY 2019

NOTE : * If the thesis is CONFIDENTIAL or RESTRICTED, please attach a thesis declaration letter.

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.

____

(Student's Signature)

Full Name : OOI YOONG KAM

ID Number : AA15161

Date : 31 May 2019

ANALYSIS OF STEEL STRUCTURE
FOR 5-STORIES
APARTMENT USING ANSYS

OOI YOONG KAM

Thesis submitted in fulfillment of the requirements
for the award of the
B.Eng (Hons) in Civil Engineering

Faculty of Civil Engineering and Earth Resources

UNIVERSITI MALAYSIA PAHANG

MAY 2019

ACKNOWLEDGEMENT

First of all, I am truly feeling 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.

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 steel 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 steel 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 steel 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 besi 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 besi 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 besi 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

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

Steel framing is one of the basic building techniques in construction field. Beams and columns are the most important component in steel framing. This is because they enable to give out the shape and supports to the structure of the building. Compare to other materials such as timber, steel has a lot of advantages. Hence it becomes one of the important materials in construction field. Despite that, it is uncommonly to apply steel framing in Malaysia as outdated of local technology and high cost of imported materials.

In construction industry, there are various types of frame. For instance, rigid frame is often used in multi-storeys building. More than 90% of non-domestic buildings' materials are steel which can show steel is dominant in this construction (Davidson & W.Owens, 2012). Steel is chosen as one of the materials in this construction is because they are light, long-span and great in durability. Other than that, it is erected rapidly that save a lot of times compared to concrete. Besides, it can save more cost since the amount of steel used in portal frame is lesser compared to other type of frame (Deakin, 2004). These allows architect to design the construction building with more pleasing appearance and economical within its cost budget. Rigid frame is the best choice for construction building as it has better stability against lateral actions. (Davidson & W.Owens, 2012).

The 3D rigid frame (15.75m height and 23.1m width) will be analyzed in this research. Wind load has to be considered in this research due to it is a high rises construction building that it sustains the wind load at ultimate and service limit state.

1.2 PROBLEM STATEMENT

Analysis of 3D steel frame is harder compared to 2D steel frame. The 3D steel frame is complicated structure as it is connected by a few of layers braced together. Therefore, it needs more times and costs in analyze and design the 3D steel frame. In civil software, ANSYS is the one of the software that can analyze the complicated structure easily. Hence, it saves more times and cost in designing a structure.

The characteristics behaviors of the steel frame are modelled and stimulated by using ANSYS + CIVILFEM 12.0 software. Deformed shape, maximum deflection and various of checking are done by applying Eurocode 3 in this research.

1.3 OBJECTIVE

Objectives are the important role in the research because it can lead researchers to meet their goals in order to accomplish the research perfectly. Furthermore, it enables researchers to comprehend and refresh themselves of the works that have to accomplish in the research. Next, the main objectives of this research are:

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

1.4 SCOPE OF STUDY

This research is conducted by using ANSYS + CIVILFEM 12.0 to analyze and model 3D steel frame. The loading that includes weight of brick walls, slabs and beams are calculated by following the Eurocode 1 then acts as force on beams. Plus, wind load is applied on the only one side of the structure thus calculated by following the Malaysia Standard. Last but not least, deformed shape, deflection and Eurocode 3 checking are done in the post-processing step.

1.5 EXPECTED OUTCOME

This research claims to determine the structural behavior of the 3D steel frame. Next, it checks all the codes of 3D steel frame. Then, the deflection, deformation, compression and tension checking, and lateral torsional buckling checking according to Eurocode 3.

1.6 SIGNIFICANCE OF STUDY

The information about the procedure and outcome of analyzing 3D steel frame by using ANSYS + CIVILFEM 12.0 is a significance attempt in this research. Through this research, it able to determine the maximum moments, deflection and deformation. Hence, this can ensure the building is safe before construction starts. Thus, it can save a lot of costs by verifying the appropriate dimension of the steel frame. So, it is utmost to understand and learn steel structure since steel frame is commonly used in advance countries. In conclusion, this research is very important and meaningful to the field of civil engineering.

CHAPTER 2

LITERATURE REVIEW

2.1 CONTINUOUS FRAME

2.1.1 Introduction to Continuous Frame

Continuous frame is the structure that made up of columns and beams. Continuous frame also named as rigid frame. Beams and columns are connected together by moment resisting connections. In frame analysis, the shear forces and beam end moments are transfer to the column through connections. This frame has a great stability against the lateral and vertical loads without any addition of bracing system. Moreover, this frame has satisfactory lateral stiffness against sway when the horizontal loads are added on it. The connections have to be rigid in the condition of the frame to be continuous. (W.Owens, Jan 2016).

2.1.2 Design of Continuous Frame

The behavior of the frame conducts the practical design of continuous frame for the multi-storey building. There are varies types of continuous frame for instance entire frame continuous and discrete continuous frame are the common types in the field of civil engineering.

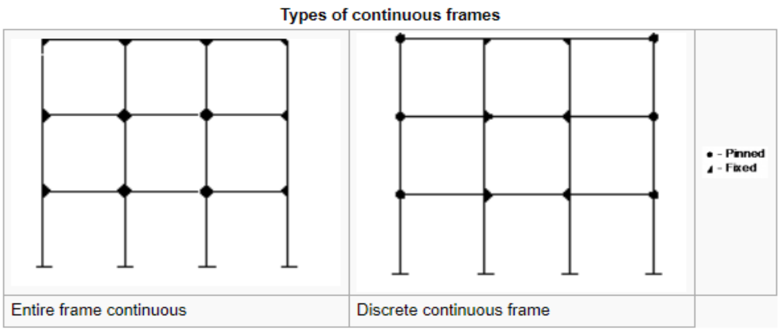

Figure 2.1 Types of Continuous Frames (Steel Construction, 2014)

For the entire frame continuous, it can be continuous throughout the whole frame since the joints are rigid. Whereas, for the discrete continuous frame, it only can be continuous at the discrete part of the frame. These two types of continuous frame that shown in Figure 2.1 enable to provide sufficient lateral resistance due to lateral loading (Steel Construction, 2014).

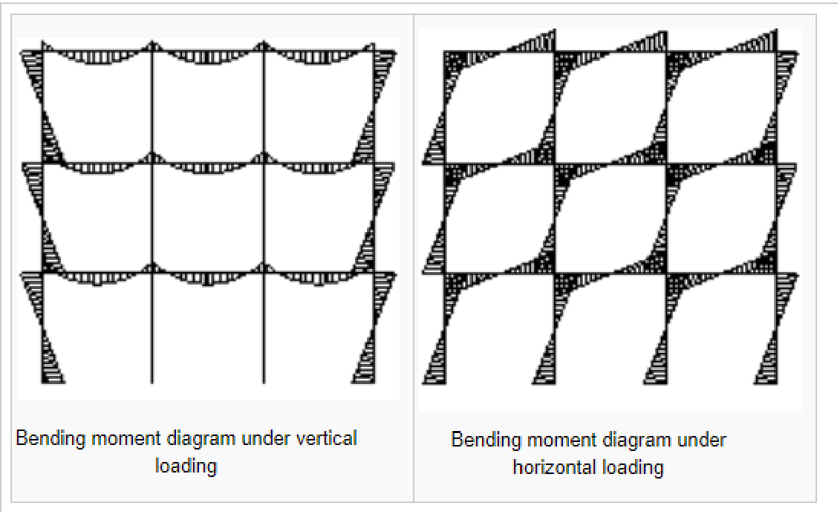

Figure 2.2: Bending moment diagram under vertical and horizontal loading

The trial beams and columns sizes have to be tested before the design and analysis of the structural frame are carried out. Bending moment acting on the beams (vertical loading) and columns (horizontal loading) are shown on the Figure 2.2. The appropriate behavior of the structural frame at serviceability limit state (SLS) can be confirmed once the determination of beams and columns section sizes is done (Steel Construction, 2014).

2.1.3 Frame Analysis at Ultimate Limit State (ULS)

2.1.3.1 Plastic Analysis

Plastic analysis is used to analyze the structural frame. Due to the plastic hinge rotation, it allows the large redistribution of bending moment in every part of the structural frame. Plastic hinge rotation is occurred at the sections when the bending moment reaches the plastic moment or resistance of the cross section at the loads below the full Ultimate Limit State (ULS) (Steel Construction, 2014). Column section is weaker compared to beam section therefore the plastic hinges will occur in the column section (Caprani, 2009/2010).

2.1.3.2 Elastic Analysis

The elastic design is used during the frame design is affected by serviceability limit state deflections. (Ross, James, Tiku, Duoc, & Sha, 2014). Elastic analysis can be determined in different structural scale and forms through ANSYS software (Duoc, et al., 2013).

2.1.4 Actions

2.1.4.1 Permanent Actions

During design a structure, the permanent actions for instance the self-weight of structural and non-structural members should have taken into considerations. Other than that, the coatings of the structure have to be taken into considerations after execution. (BS EN 1991-1-1:, 2002). Self-weight of the cladding and steelwork which can be acquired from the Eurocode or manufacturer also have to be calculated as they are part of the permanent actions for steel structure (Steel Construction, 2014).

2.1.4.2 Variable Actions

Variable action is also named as live load or imposed load. This load can change according to the usage and capacity. People, furniture, moveable partitions, snow or roof loading are considered as imposed load (BS EN 1991-1-1:, 2002). According to the building, minimum and maximum imposed loading are determined that provided in Eurocode.

2.1.4.2.1 Wind Actions

The wind actions are tremendously hard to determine. This is due to the topography and location of site are the one of the factors that affects the magnitude of the wind actions. Wind actions often ignored for the low rises multi-storey buildings. The standard is provided in Eurocode to ensure the wind analysis of the buildings can be done (M.T. & Jose, 2014). Nevertheless, local engineers use the Malaysia Standard which is the guidelines to determine the wind actions whenever the site is located in Malaysia area.

2.1.5 Connections

In multi-storey buildings, moment resisting connections commonly occurred in beam-to-beam and beam-to-column connections. Full depth end plate connection and extended end plate are the two types connections that usually used.

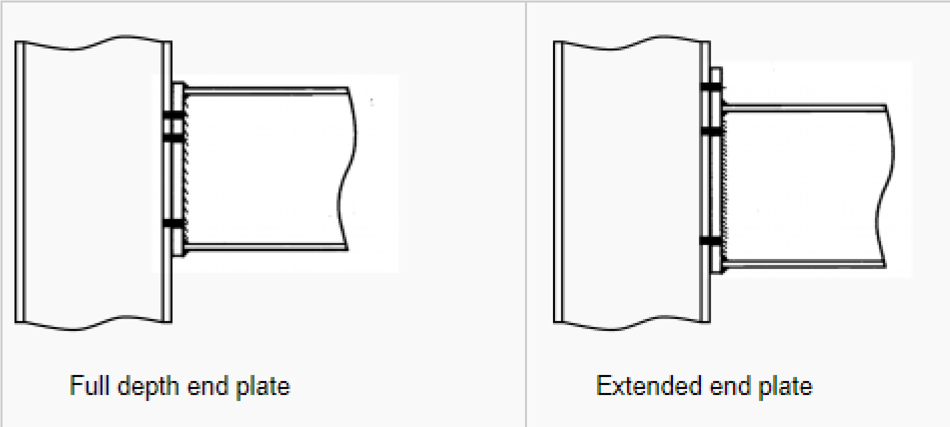

Figure 2.3: Commonly used moment resisting connections

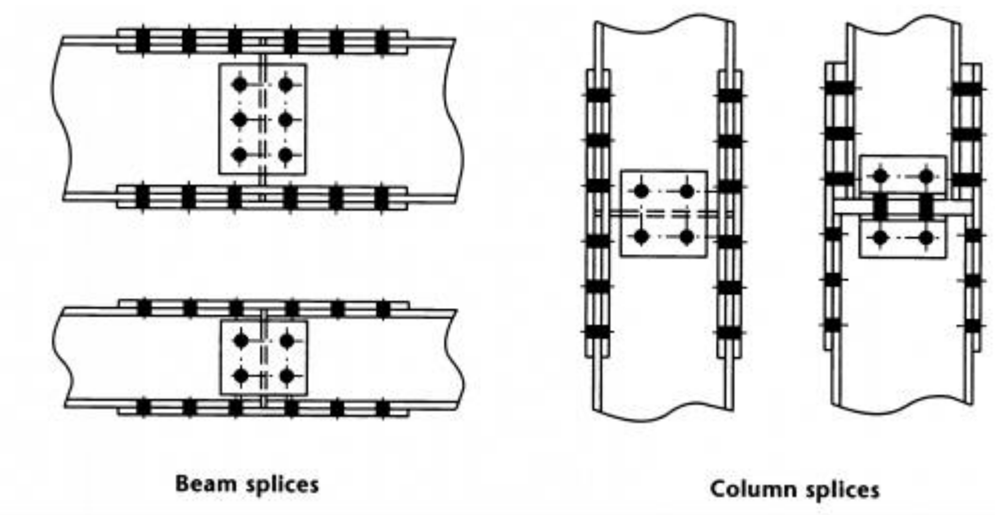

Figure 2.4: Typical bolted cover plate splices

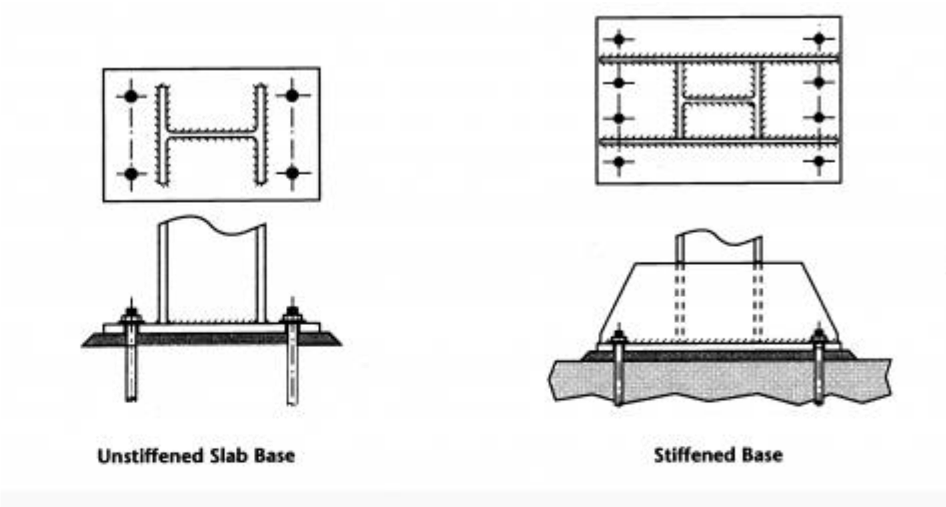

Figure 2.5: Typical column base

Column base is able to transfer axial force and moment between the concrete substructures and steel members. Unstiffened slab base commonly used in construction due to lower cost than the stiffened base (Steel Construction, 2014).

Economic factor is one of the factors in choosing the connections type in steel structural. Hence, bolting is recommended to use in connections compared to welding. This is because bolting reduces the cost of labor and also fasten the speed in site erection (Steel Construction, 2014). The frame stability depends on the rotational stiffness. Therefore, connection stiffness needs to be determined by using Eurocode 3 to ensure the connection design matches the requirements (BS EN 1993-3-8:, 2005).

2.2 ANSYS SOFTWARE

2.2.1 INTRODUCTION

In finite element modelling, ANSYS is a powerful software which can work out various mechanical problem. For instance, both linear and non-linear static, dynamic structural analysis, fluid problem, heat transfer, acoustic and electromagnetic problems and so on (University of Alberta - ANSYS Tutorials, 2010).

In order to run an analysis of a structure, three stages need to be carried out. In first stage (pre-processing step), types of cross section, elements types, and members properties must be clarified. Plus, nodes and elements have to be accomplished in order to achieve the model of the structure. Next, the second stage is solution stage that the types of support of the structure and the loadings applied on the members are checked and verified. Last but not least, the post-processing stage is the third stage. The outcome of the analysis is generated by using ANSYS software in this stage.

2.2.2 Continuous Frame ANSYS Analysis

The structural behavior of the continuous frame is analyzed by ANSYS software. Moreover, this software also can analyze details of beams, columns and connections. Furthermore, it able to generate the simulation of the continuous frame under different type of condition for instance analysis of continuous frame under fire conditions. ANSYS is a non-linear finite analysis tool (P.J., R.P., M.W., & A.H., 2008). The actual behavior of the members of the continuous frame can be determined through ANSYS software (Elsayed, Mohamed, Hamdy, & Mohamed, 2010).

2.3 ANALYSIS METHOD

2.3.1 Finite Element Analysis

Finite element method is applied to analyze the whole structure by subdivide the structure into finite-size elements and connected at nodes thus easier for calculations. The more amount of the finite elements the more accuracy of the outcome. This method provides a high precise in the estimation of the stresses in all the members. Finite element method is an extension of Rayleigh-Ritz method. This method is able to overcome the obstacles when the structure is big and complex (G Lackshmi, 2009).

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

(i) Thermal analysis

This analysis able to evaluate the temperature distribution in the members. There are two types of operations which are linear and non-linear. Linear operation is the steady state or temporary heat transfer by convection and conduction whereas the non-linear operation is the radiation. Thermal effect is important by assuming temperature range which depends on the location of the site and thermal coefficient of expansion for members.

In the research, the behavior of frame structure exposed in fire condition is conducted. Through this research, the temperature distribution of the frame structure exposed to high temperature that effects the response of the members is acquired successfully in thermal analysis (Lenka Lausovaa, 2017).

(ii) Structural Analysis

The stress and displacement for the framed structure are determined through the finite element method. Other than that, the finite element method able to analyze the structure under many types of loads for instance gravity, pressure, wind and temperature. Plus, the dynamic load may also be considered. In the research, the high rises building is analyzed by using finite element method. ANSYS software is used in this research to carry out the value and distribution of the internal forces, stresses and displacements (Jing Ji, 2012).

(iii) Probabilistic Analysis

This analysis is designing a structure frame by assuming the probability of failure instead of safety factor. It deals with the effects of random variable on the structural or the structural members during the design phase that related to the reliability and stability. The input parameters are given in a random probability distribution and categorized by different distribution types. Then, the outcomes of these probabilistic analyses are conducted through various analysis loops in ANSYS software (Tvrda, 2017).

2.3.2 Response Surface Method

Response surface methodology is a method that apply mathematical and statistical ways to design those experiments. The optimize output parameter that affected by a few of random input parameters could be conducted for the structure. This method is one of the alternative methods which produce a fast and efficient computation of response of the large and complex structure. Moreover, it helps to solve the problems in finite element method that acquires carry out various runs with large number of input variables (Goswami, Chakraborty, & Ghosh, 2013).

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

The methodology of the research is to discussed in this chapter thoroughly by presenting the steps of analyzing the multi-storey building formed by continuous frame. This research is about the finite element analysis and code checking by Eurocode 3 of 3D steel frame. ANSYS + CIVILFEM 12.0 software is used to analyze the structure of 3D steel frame. A very little cost is needed for this software in order to do the simulation of this research. The material, cross section and element type of the steel structure are 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 steel 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

Pre-processing process, solution process and post-processing process are the three steps to analyze the 3D steel frame. In the pre-processing process, the 3D steel frame is modelled by defining the material, cross section, element type thus creates the nodes and elements to form the 3D steel frame. Next, in the solution process, the type of support is allocated to the structure and the loads act as pressure are inputted on the beams and columns of the structure. Then, the analysis of the 3D steel frame is started before proceed to the last step. Last but not least, outcome that noticed in the objective is conducted in the post-processing process. The flow of this research is shown in Figure 3.1.

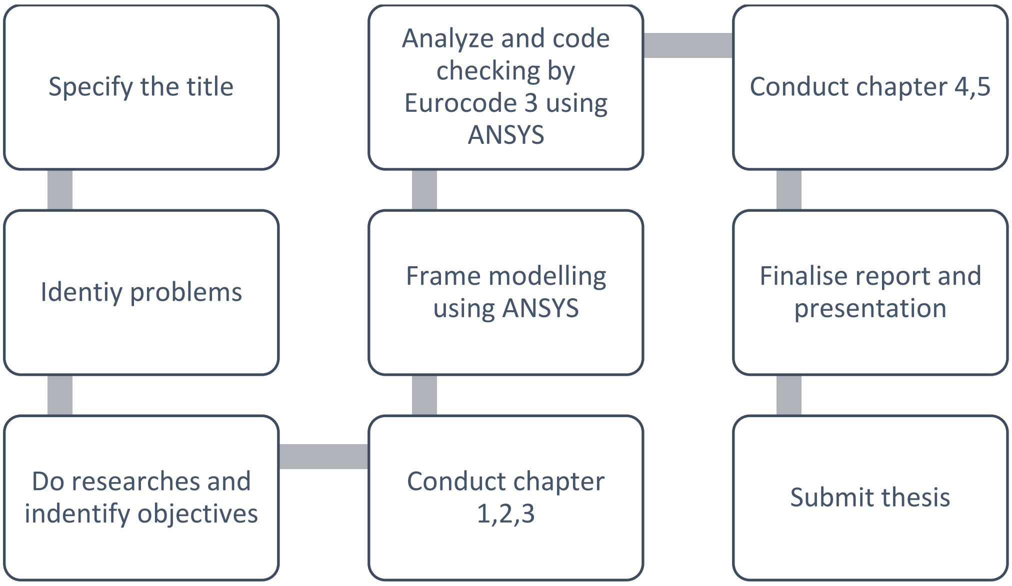

Figure 3.1: Flow chart of Research Process

3.2 Preprocessing

In this step, the input data of 3D steel frame is determined according to the Eurocode 3 (EN 1993-1-1:2005). A 3D model of steel frame is created in this step.

3.2.1 Entering Title

The title is created for the analysis. To specify a title, it is done by clicking the Utility Menu Bar at the top of the window then select File and click on the Change Title next type in the name of the title which is “Steel Frame”. So, the title will publish at the bottom left corner. In order to check the model by Eurocode 3, the CIVILFEM need to be activated. From the Main Menu, select CIVILFEM and click OK to activate it as shown in Figure 3.2 below.

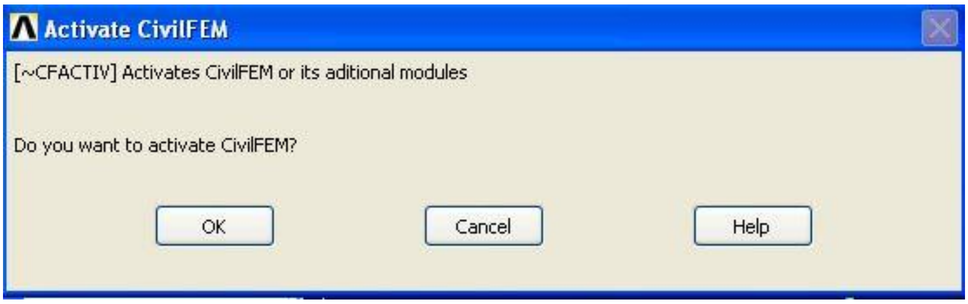

Figure 3.2: Activate CIVILFEM

3.2.2 Set Codes and Units

In this research, the Eurocode 3 is used as active code in checking the steel frame. Click the Main Menu of ANSYS and select Civil Setup to set the code for this research. Eurocode 3 (EN 1993-1-1:2005) is selected for the code checking of the steel frame. Then, the International System Units is the choice of the system units in this research.

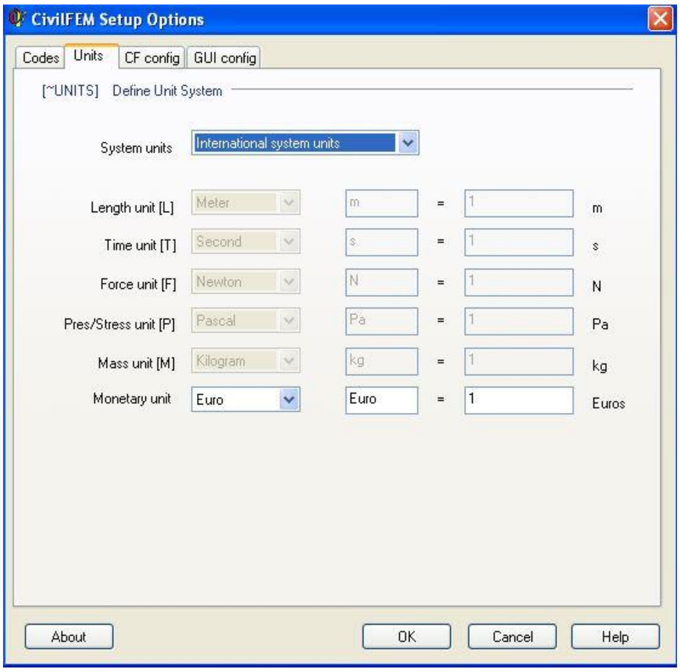

Figure 3.3: CIVILFEM setup options for units

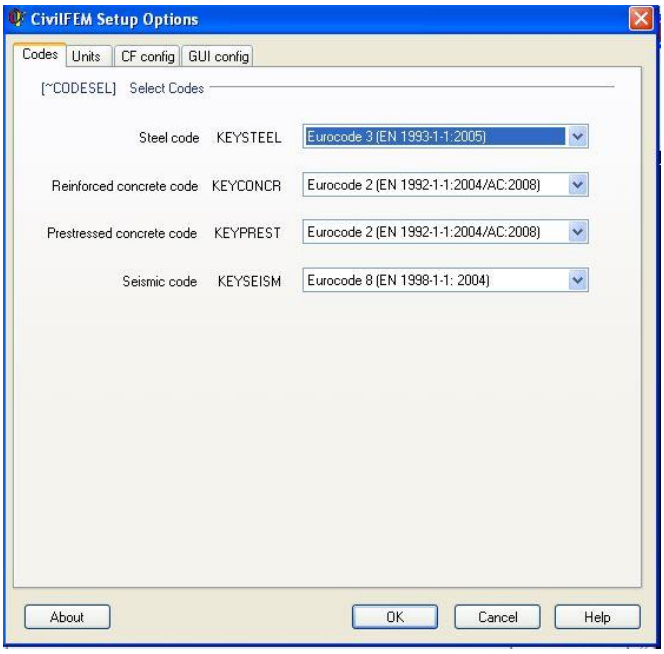

Figure 3.4: CIVILFEM setup options for codes

3.2.3 Defining Element Types

Through this software, the element type of the structure can be determined. Civil Preprocessor is selected from the Main Menu followed by Element Type and Civil Beam. 3D Lin. Beam 188 is selected as shown in Figure 3.5 below.

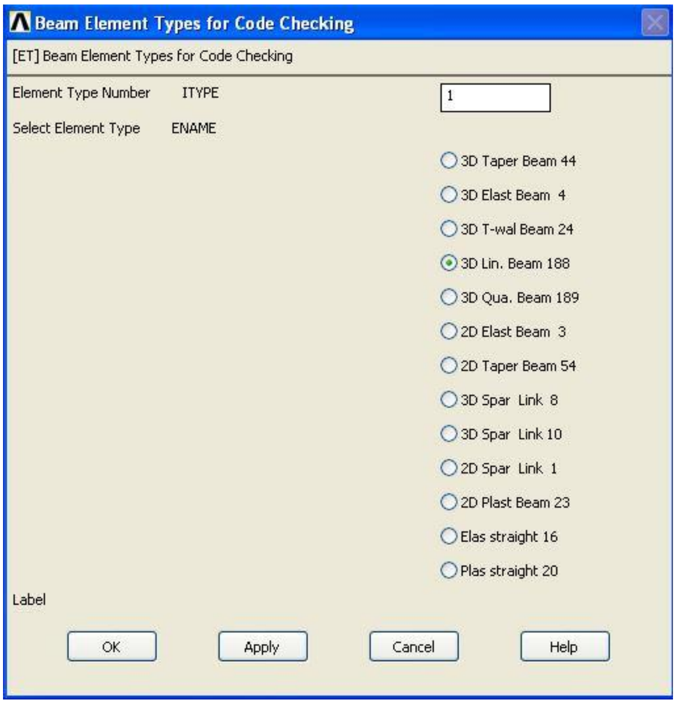

Figure 3.5: Selecting element type

3.2.4 Defining Material

To verify the materials used in this research, Materials under Civil Preprocessor from the Main Menu is selected. After Material Browser appeared, New is clicked. Various types of material are available to choose in the box. Hence, structural steel with FE355 under EC3 is selected as the material of this research.

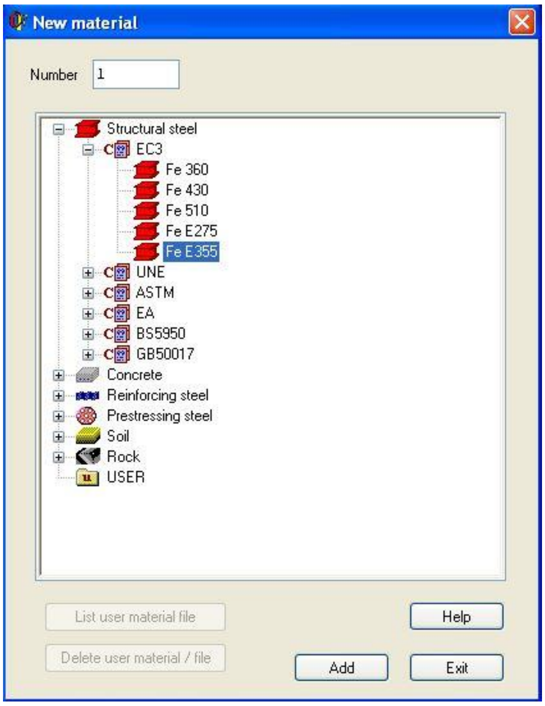

Figure 3.6: New Material

3.2.5 Defining Section

Cross Section under Civil Preprocessor is clicked from the Main Menu. To enter the Hot Rolled Library of CIVILFEM, the Hot Rolled is selected. For beams, UB 914 x 419 x 388 is selected. Whereas, for columns, UC 356 x 406 x634 is selected.

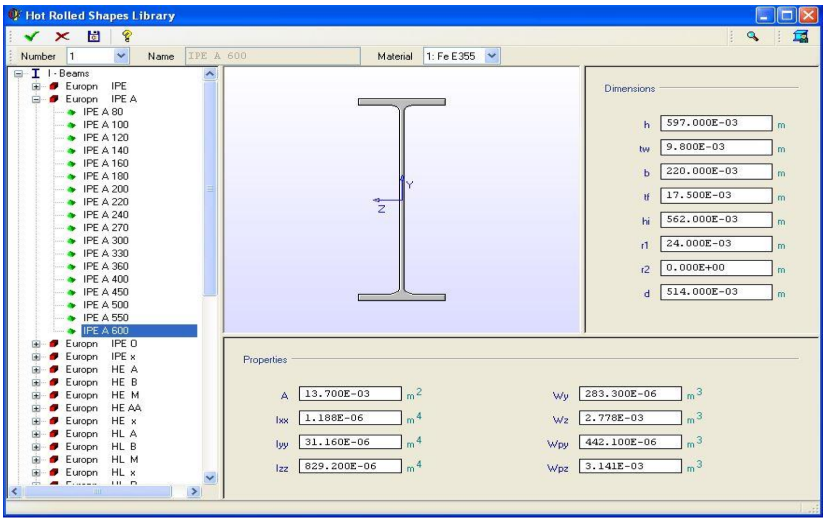

Figure 3.7: Steel cross section

3.2.6 Defining Member Properties

The aim of defining member properties is to determine the compression buckling for the compression members. Member Properties which is located at the Civil Preprocessor is selected. New property is clicked to define beam and column in the window of Member Properties. Length of the beam is inserted then the “LatBuck” is unticked and “-Z” in “ChckAxis” is chosen. The steps above are repeated in defining column. The members properties are shown below the Table 3.2.

Table 3.2: Member Properties

No.	Name	Length (m)
1	Beam	6.00
2	Column	3.15
3	Beam 1	10.00

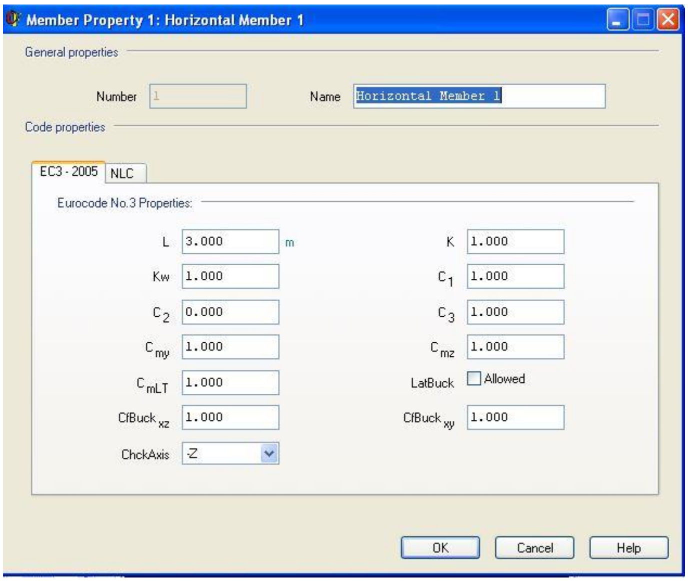

Figure 3.8: Member Properties

3.2.7 Defining Beam and Shell Properties

To define ANSYS real constants, the beam and shell properties is applied. In the window of the beam and shell properties, the new beam is selected. Beam 188 is chosen as the Ename. Next, “1” is chosen in the member properties number and top of the cross-section diagram. After defining, the tick is clicked. These steps are repeated in defining the column’s property.

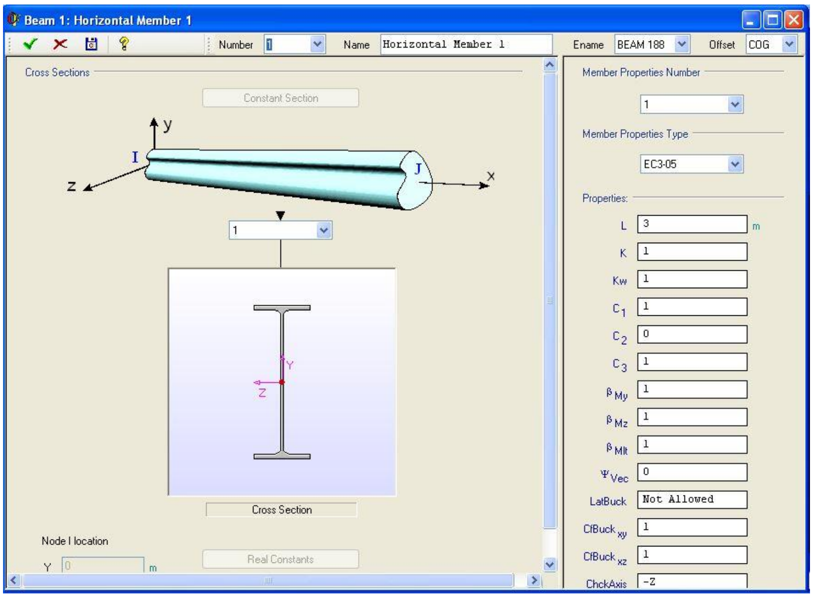

Figure 3.9: Beam

Table 3.3: Beam & Shell Properties

No.	Name	Member Properties Number
1	Beam	1
2	Column	2
3	Beam 1	3

3.2.8 Defining Nodes and Elements

Preprocessor under the Main Menu is clicked as to define the keypoints and element s of the structure. Modelling, Create and Keypoints are clicked accordingly. In the window of Create Keypoints in Active Coordinate System that shown in Figure 3.10, the “1” is inputted into the Keypoint Number then 0,0,0 is inputted as the first coordinate of the structure. These steps are repeated until all 48 keypoints are inputted. The keypoints with their coordinates are shown in Figure 3.11.

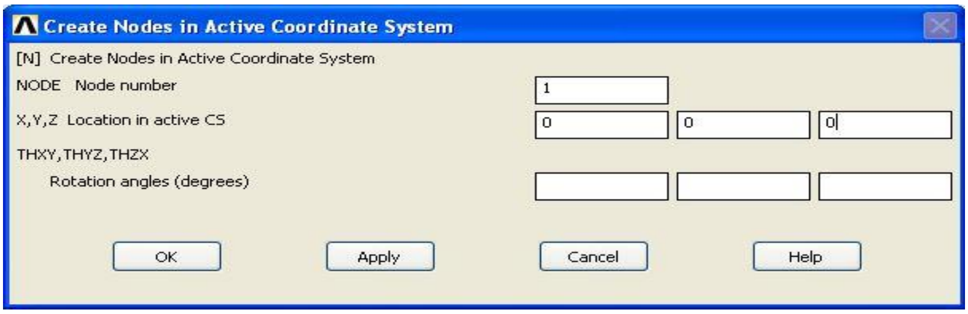

Figure 3.10: Create Nodes in Active Coordinate System

Nodes	Coordinates			Nodes	Coordinates		
1	0	0	0	25	0	0	10
2	6	0	0	26	6	0	10
3	12	0	0	27	12	0	10
4	18	0	0	28	18	0	10
5	0	3.15	0	29	0	3.15	10
6	6	3.15	0	30	6	3.15	10
7	12	3.15	0	31	12	3.15	10
8	18	3.15	0	32	18	3.15	10
9	0	6.3	0	33	0	6.3	10
10	6	6.3	0	34	6	6.3	10
11	12	6.3	0	35	12	6.3	10
12	18	6.3	0	36	18	6.3	10
13	0	9.45	0	37	0	9.45	10
14	6	9.45	0	38	6	9.45	10
15	12	9.45	0	39	12	9.45	10
16	18	9.45	0	40	18	9.45	10
17	0	12.6	0	41	0	12.6	10
18	6	12.6	0	42	6	12.6	10
19	12	12.6	0	43	12	12.6	10
20	18	12.6	0	44	18	12.6	10
21	0	15.75	0	45	0	15.75	10
22	6	15.75	0	46	6	15.75	10
23	12	15.75	0	47	12	15.75	10
24	18	15.75	0	48	18	15.75	10

Figure 3.11: Keypoints with Coordinates

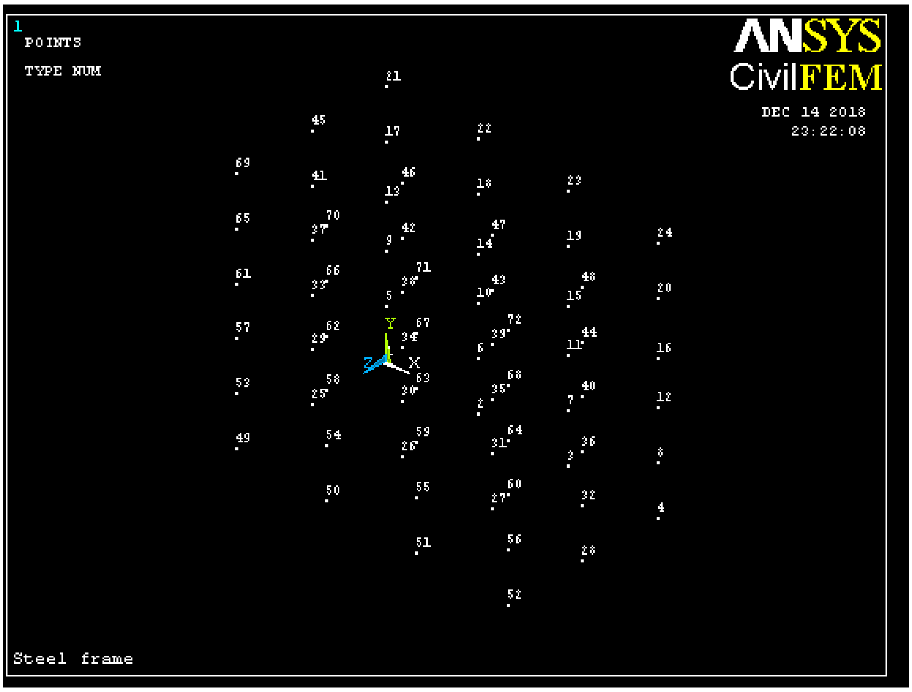

Figure 3.12: Total of 48 Keypoints

3.2.9 Creating of Model

Before meshing the structure, the keypoints have to connected with each other by using Line Tools. Meshing tool is used to differentiate the element of each line since they have their own length and properties. Mesh tool is clicked under the Preprocessor and Meshing. Global in the element attributes is selected. The section “1” is selected for beam whereas “2” for column in the meshing attributes window. Size changed to 2 and Smart Size is ticked. Last but not least, Mesh is clicked. The meshed lines with their own element attributes in shown in the figure below.

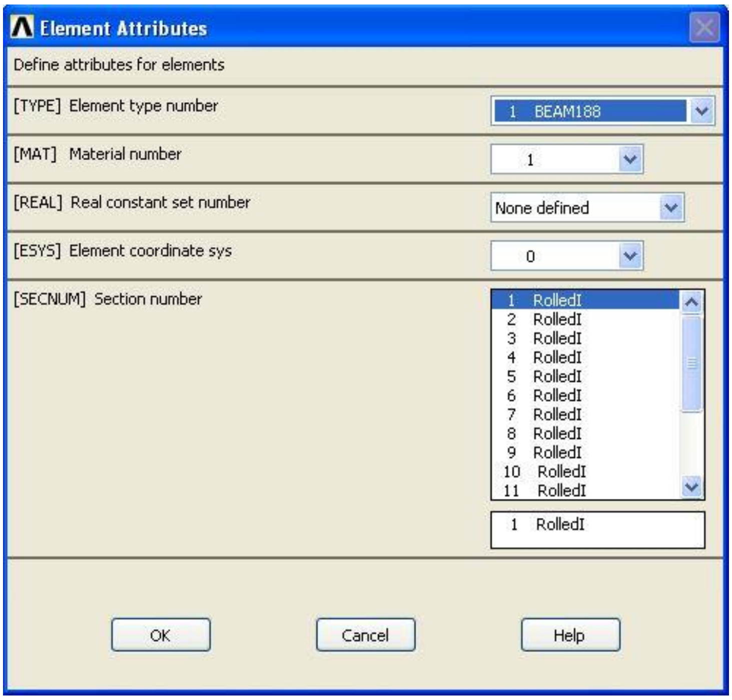

Figure 3.13: Element Attributes

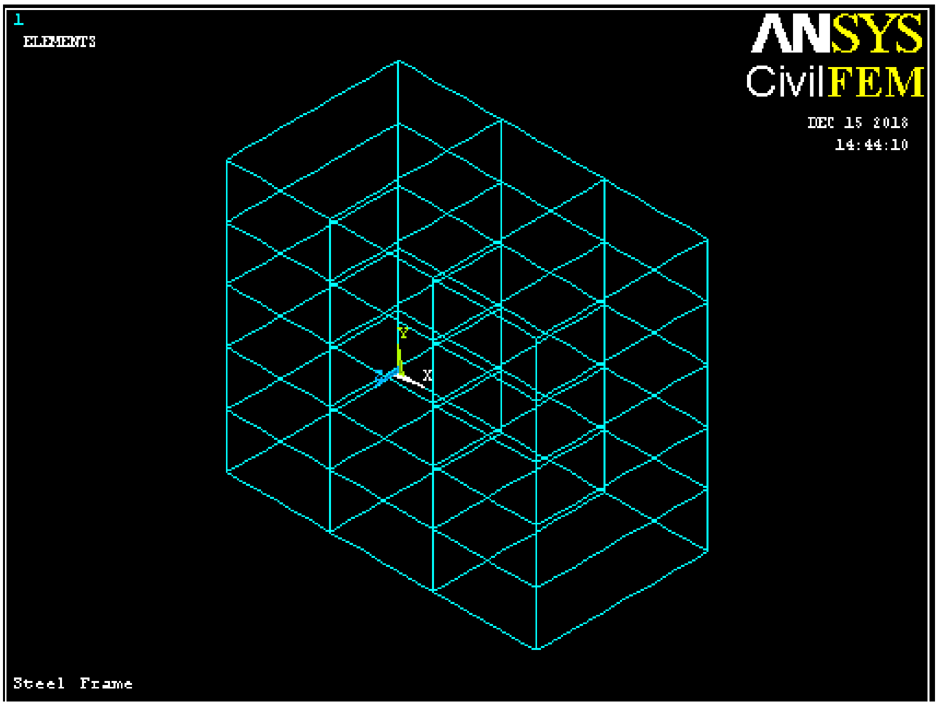

Figure 3.14: Model of 3D steel frame

3.3 Solution Phase

Forces and setting of constraint are determined in the structure in this stage. Then, the analysis is solved. Hence, it is the utmost step in analyzing the steel frame before proceed to the step 3 which is post-processing step.

3.3.1 Define Analysis Type

Static is selected in the New Analysis under the Main Menu. OK is selected to confirm the choice.

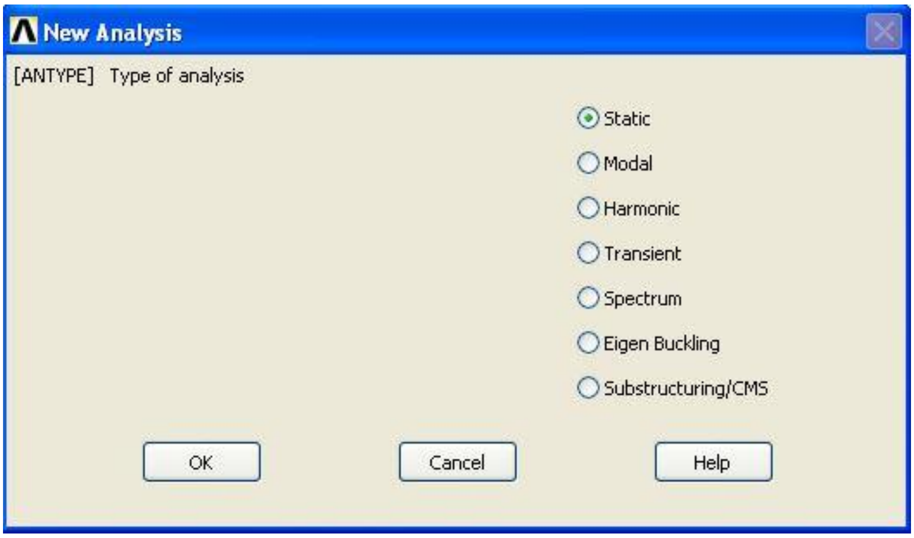

Figure 3.15: Type of Analysis

3.3.2 Apply Constraints

Solution, Define Loads, Apply, Structural, Displacement and On Node are clicked accordingly. All DOF is selected for the DOFs to be constrained then 0 is inserted as the displacement value after selecting the nodes that positioned at the ground beam. The outcome of the displacement constraint applied in the steel frame model is shown in the figure below.

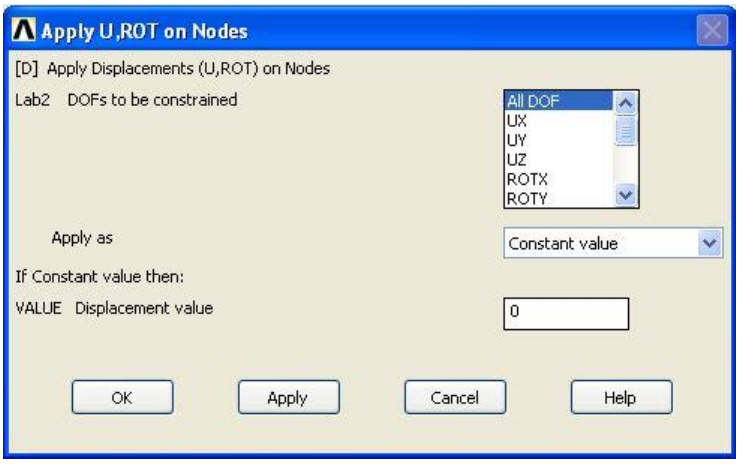

Figure 3.16: Apply U,ROT on nodes

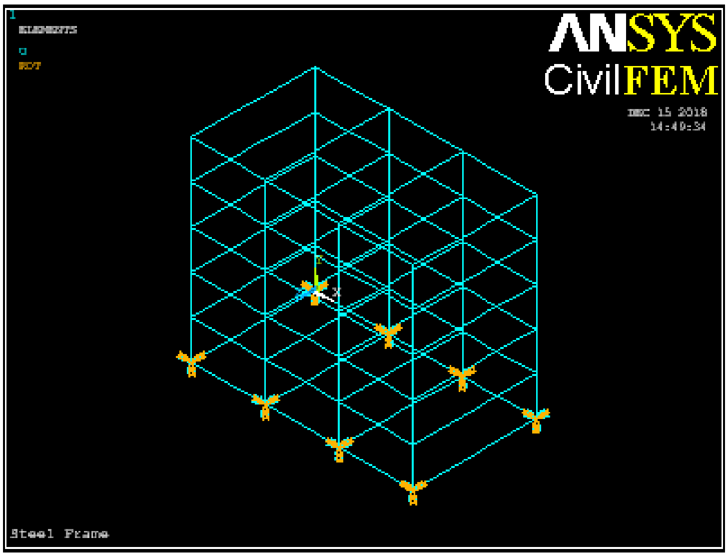

Figure 3.17: Displacement constraint applied on model

3.3.3 Apply loads

Solution, Define Loads, Apply, Structural, Force/Moment and On Nodes are selected accordingly. FY is selected as the direction of the force or moment meanwhile the value of force/moment, -20000 is inputted in order to apply the load in vertical direction. The negative sign indicates the opposite y-direction. The dead and live loads are calculated as the form of point load.

The wind load is inputted after putting the point load. The wind load is assumed acting on one side of the steel frame. The value of wind load is inserted into the pressure value at the node I. The model that applied by point and win loads is shown in the figure below.

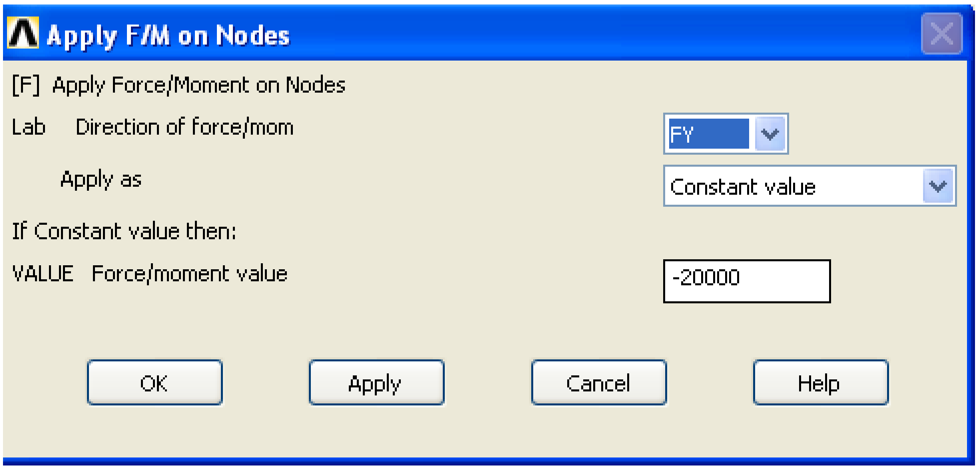

Figure 3.18: Apply Force/Moment on Nodes

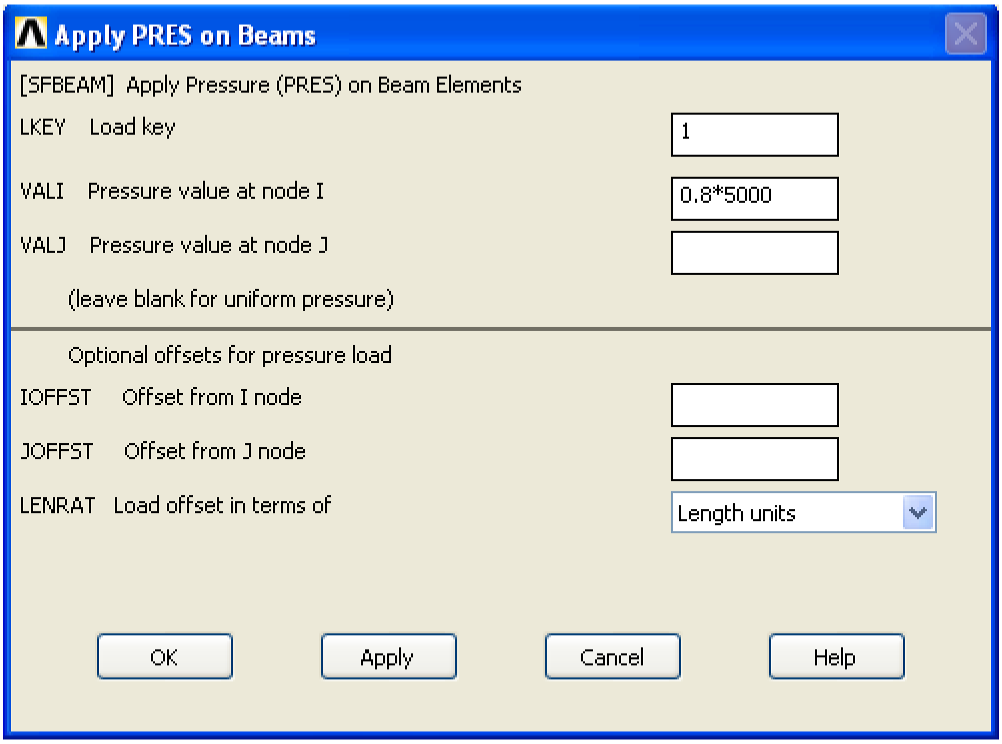

Figure 3.19: Apply PRES on Beams

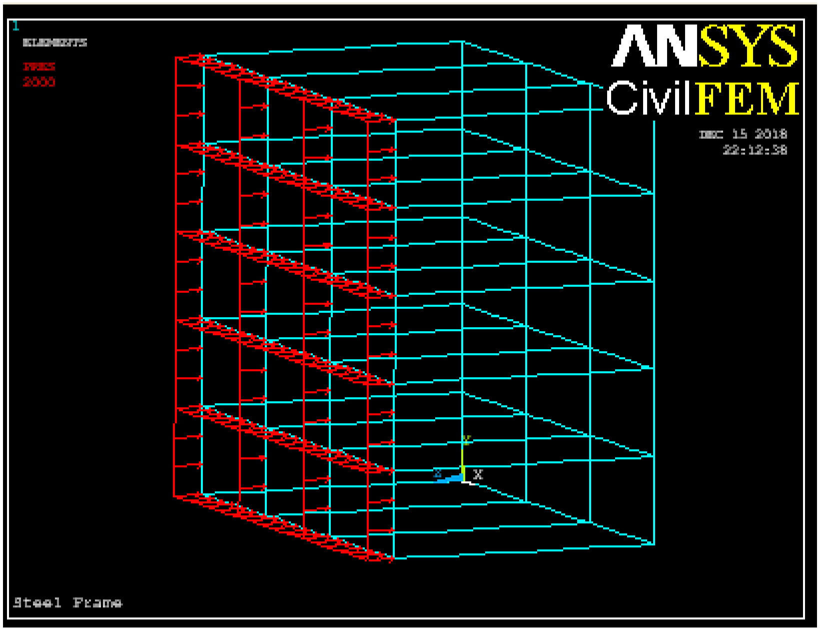

Figure 3.20: Model that applied by point and wind loads

3.3.4 Solving

Current LS under Solve is selected to start the solution of current Load Step (LS). The solution is done after OK is clicked.

Figure 3.21: Solve Current Load Step

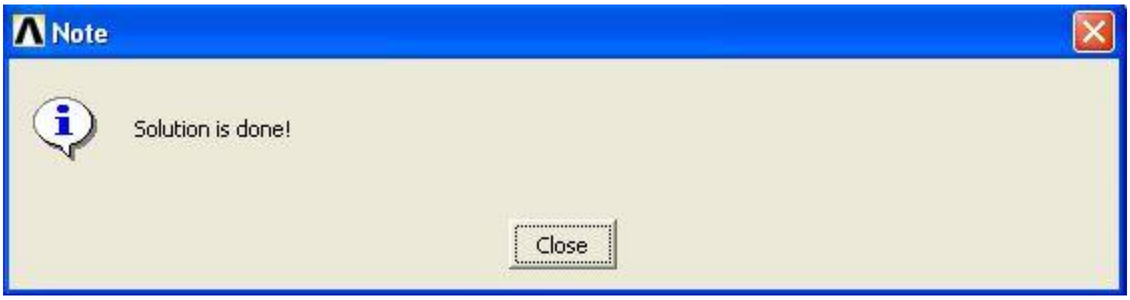

Figure 3.22: Note

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

Deformed Shape under Plot result is selected. Then "Def+Under Edge" is selected to show the deformed shape that is plotted.

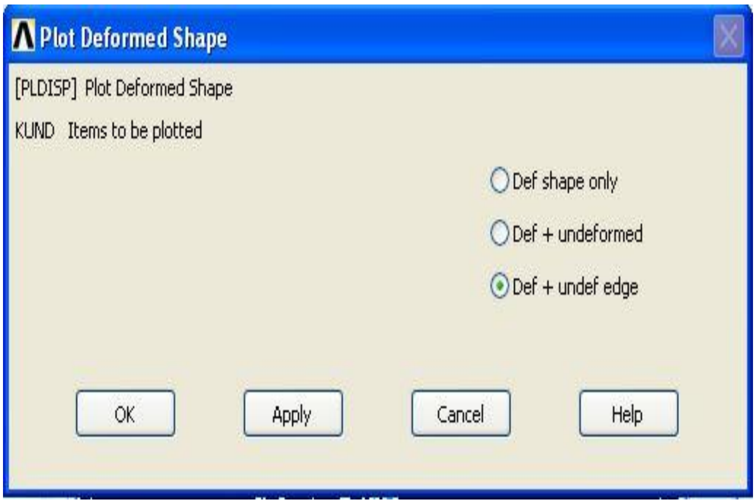

Figure 3.23: Plot Deformed Shape

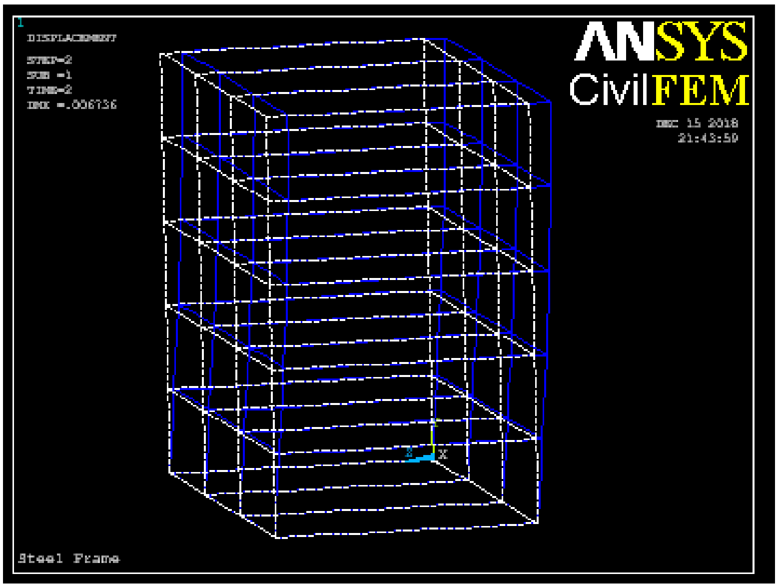

Figure 3.24: Deformed Shape of the Steel Frame

When the forces are applied to the steel frame, the shape of the steel frame is deformed. Plastic deformation would not return back to its original shape after the forces applied are removed whereas the elastic deformation would return back to its original shape. From the figure above, the structure tends to move to negative z-direction since the force are greater on that side.

3.4.2 Deflection

The General Postproc, Plot Results and Contour Plot are selected accordingly. The Nodal Solution and Displacement vector sum are selected accordingly after the window is popped out. OK is selected hence the results are plotted. Through this displacement checking, the deflection of the structure can be determined under the real condition by simulation. Critical part of the steel frame can be determined as the figure stated below.

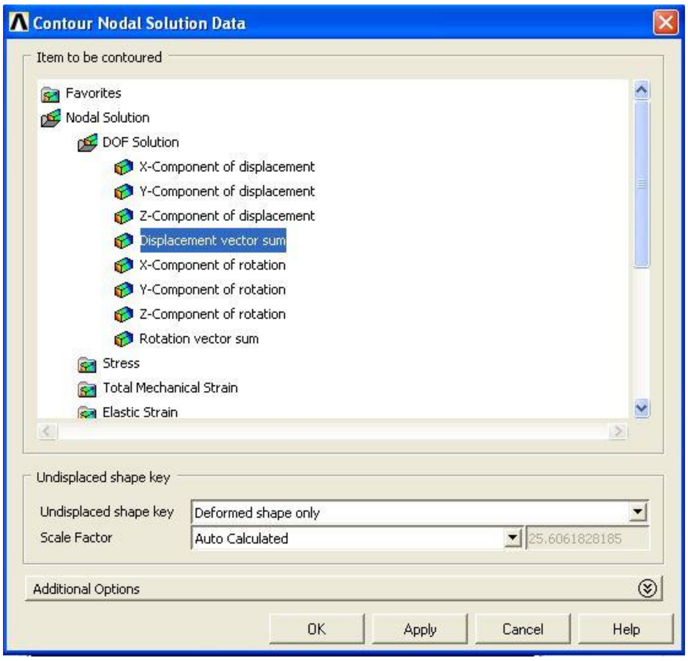

Figure 3.25: Contour Nodal Solution Data

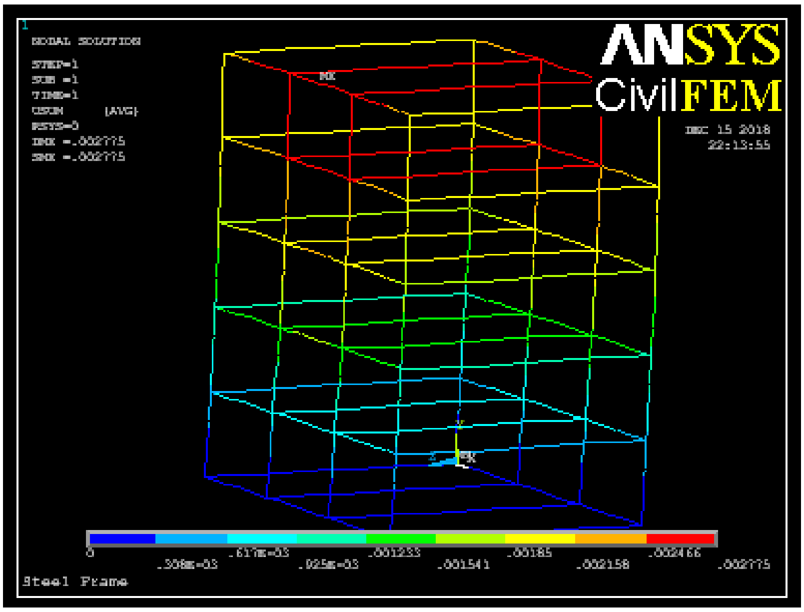

Figure 3.26: Contour Plot of Deflection

The displacement vector sum of the members is shown figure above. The highest displacement in the figure above is 2.775mm. The blue colour indicates lesser displacement. The range of displacement from the contour plot is 0mm until 2.775mm. Small displacement occurred on the most parts of the structure.

3.4.3 Read Results

This step is needed to read the results for the following checking by CIVILFEM. This can be accessed through the Civil Postprocessor, Read Results and By Load Step.

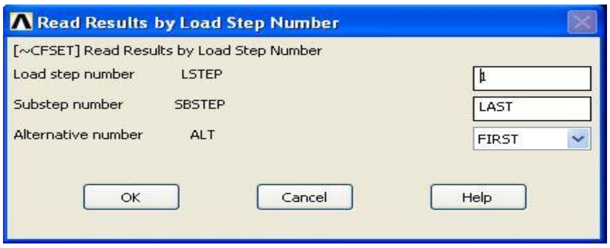

Figure 3.27: Read Results by Load Step Number

3.4.4 Forces & Moments

Axial force, shear force and bending moment are able to be plotted in this step. Beam Utilities, Graph Results and Forces & Moments are selected accordingly. Axial Force X is selected to plot the results for Axial Force. Then, these steps are repeated to plot Shear and Bending Moments.

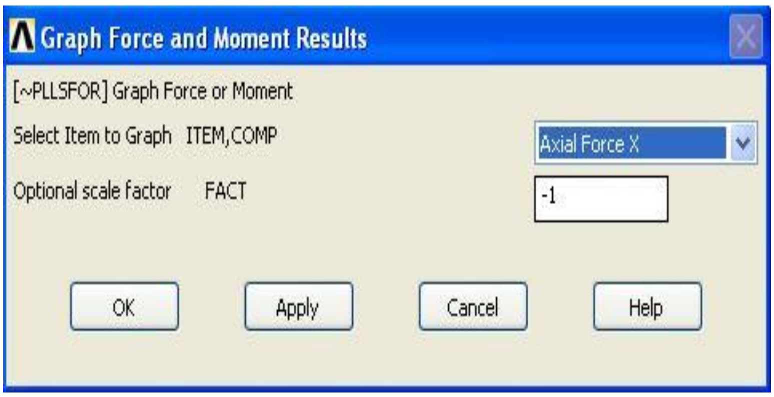

Figure 3.28: Graph Force and Moment Results

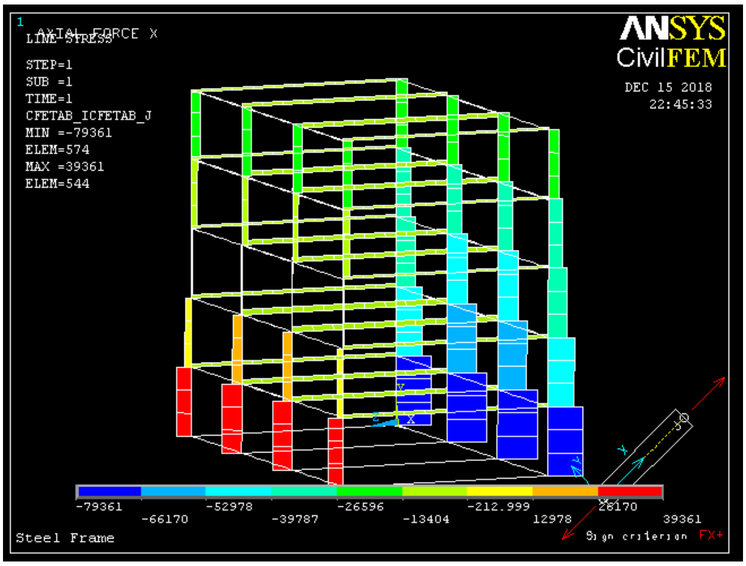

Figure 3.29: Axial Force Diagram

The axial forces in each member of the structure in the Figure 3.29. The range of the axial forces in the member is -79361N to 39361N. The contour indicates different axial forces between the ranges.

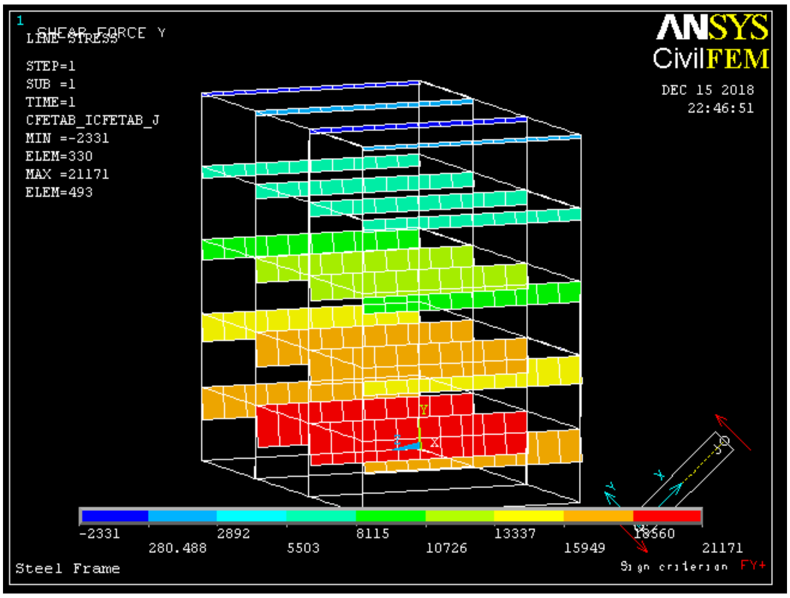

Figure 3.30: Shear Force Diagram

The shear forces in each member of the structure in the figure 3.30. The range of the shear force is from -2331N to 21171N. The contour indicates different shear forces between the ranges.

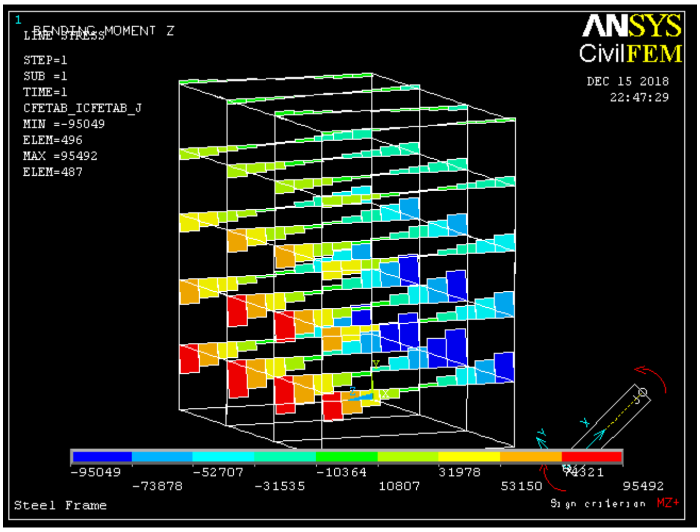

Figure 3.31: Bending Moment Diagram

The bending moments in each member of the structure is shown in the Figure 3.31. The range of the bending moment is from -95049Nm to 95492Nm. The contour indicates different bending moments between the ranges.

3.4.5 Code Checking

The utmost checking for a steel frame is code checking. Only Eurocode 3 will be used to check in this research. The Civil Postprocessor, Code Checking, Eurocode 3, Check by Code and Tension are selected accordingly. CIVILFEM -Z Axis is selected. Therefore, CIVILFEM is allowed to analyze the steel frame according to Eurocode 3. The results are plotted under the Beam Results. Plot Results is selected then Tension and Element OK is selected. OK is selected to confirm the results that have to be plotted. These steps are repeated for the compression, bending, shear, compression buckling and lateral buckling.

Figure 3.32: Check Model Results By Eurocode 3

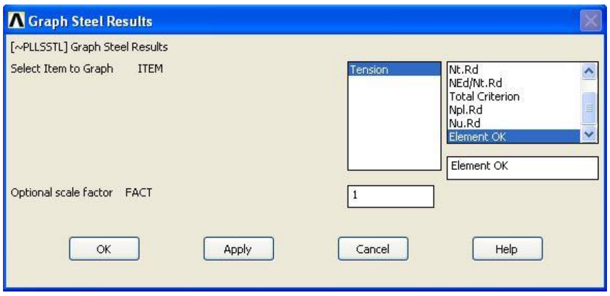

Figure 3.33: Graph Steel Results

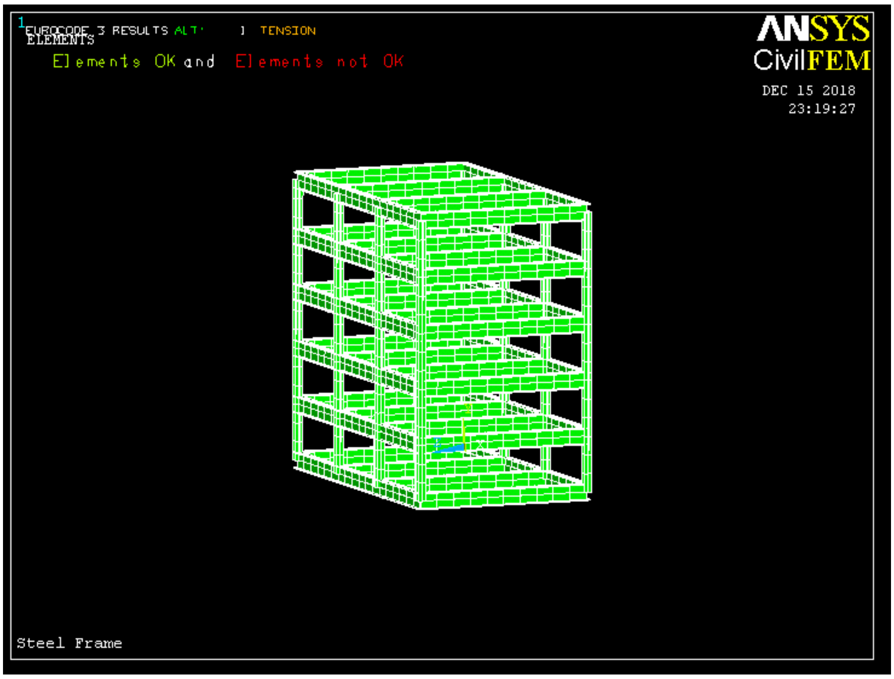

Figure 3.34: Tension Checking Results

The results of checking of tension by using Eurocode 3 in the Figure 3.34. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

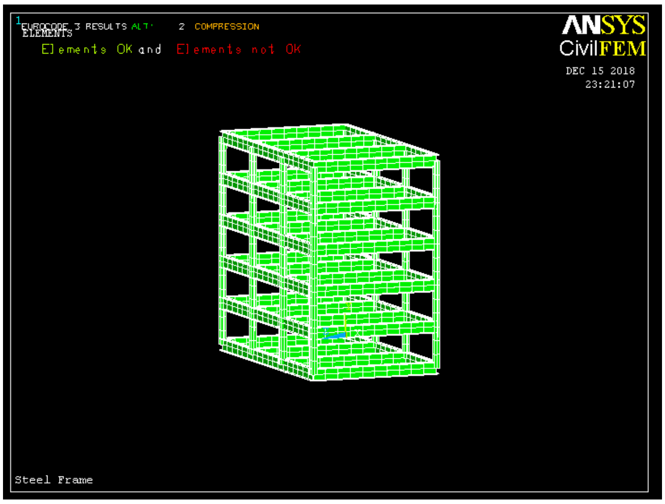

Figure 3.35: Compression Checking Results

The results of checking of compression by using Eurocode 3 in the Figure 3.35. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

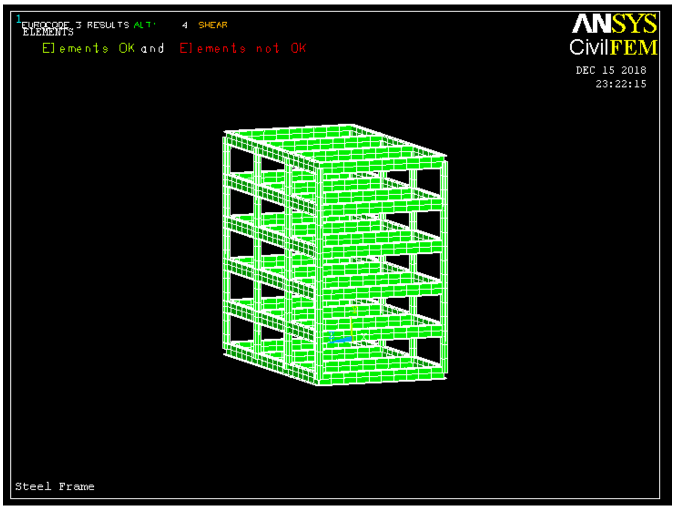

Figure 3.36: Shear Resistance Checking Results

The results of checking of shear resistance by using Eurocode 3 in the Figure 3.36. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

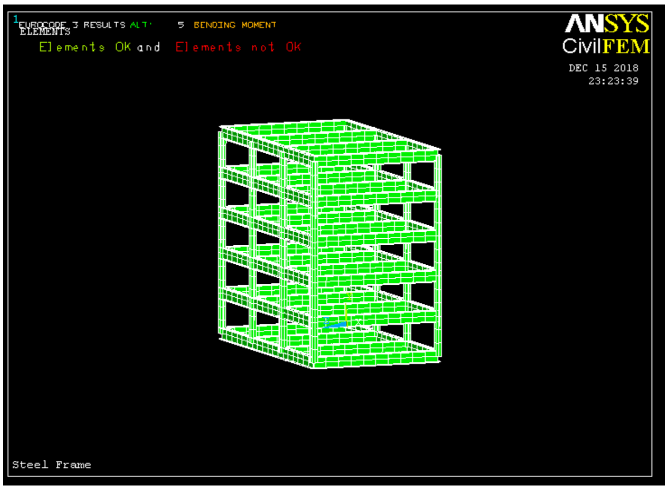

Figure 3.37: Bending Moment Resistance Checking Results

The results of checking of bending moment resistance by using Eurocode 3 in the Figure 3.37. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

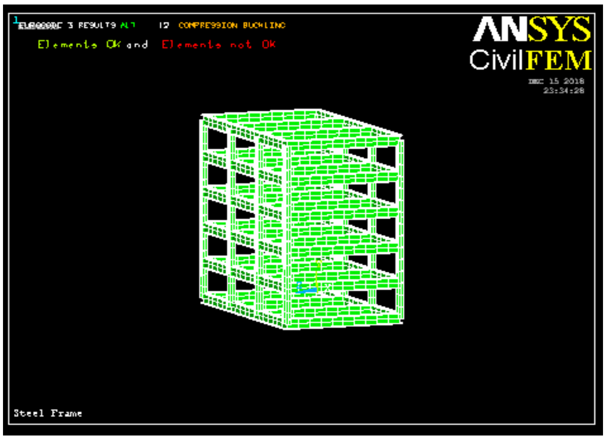

Figure 3.38: Compression Buckling Checking Results

The results of checking of compression buckling by using Eurocode 3 in the Figure 3.38. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

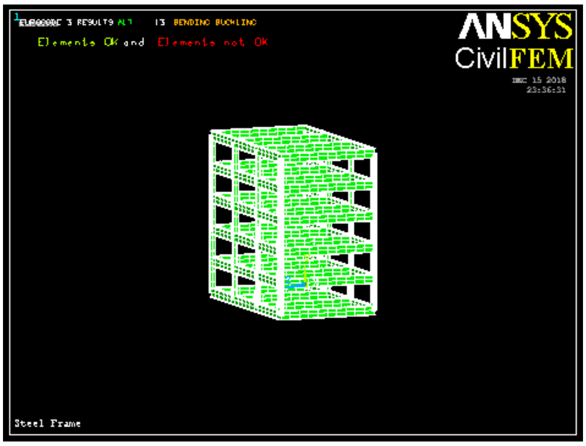

Figure 3.39: Lateral Torsional Buckling Checking Results

The results of checking of lateral torsional buckling by using Eurocode 3 in the Figure 3.39. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

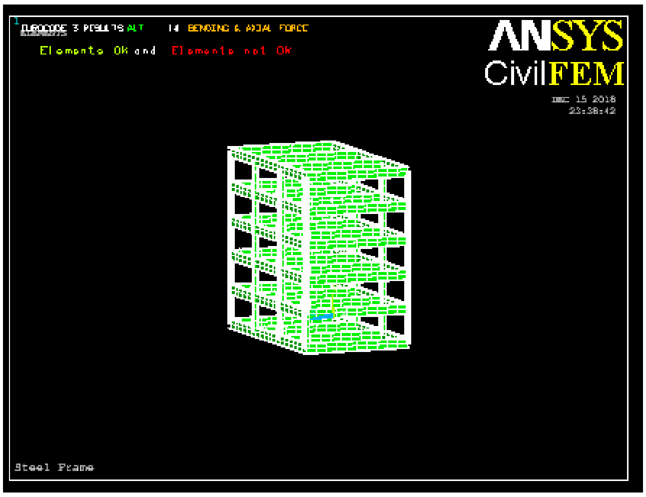

Figure 3.40: Bending + Axial Force Checking Results

The results of checking of combination of bending moment and axial force by using Eurocode 3 in the Figure 3.40. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

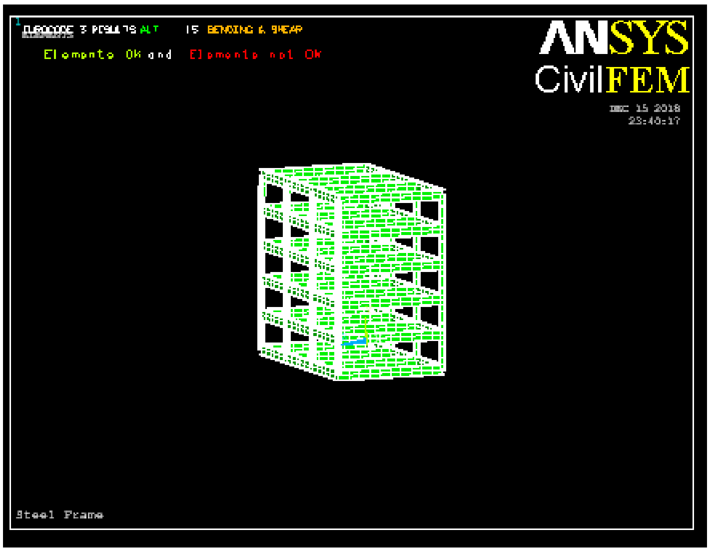

Figure 3.41: Bending + Shear Force Checking Results

The results of checking of combination of bending moment and shear force by using Eurocode 3 in the Figure 3.41. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

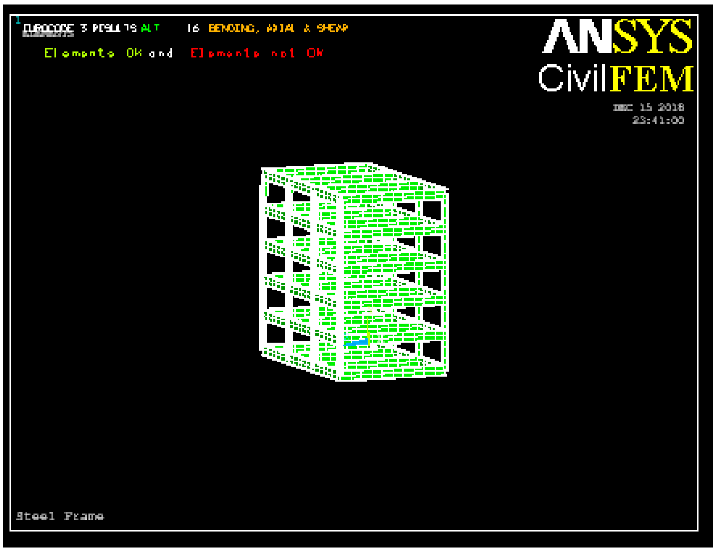

Figure 3.42: Bending + Axial Force + Shear Force Checking Results

The results of checking of combination of bending moment, axial force and shear force by using Eurocode 3 in the Figure 3.42. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

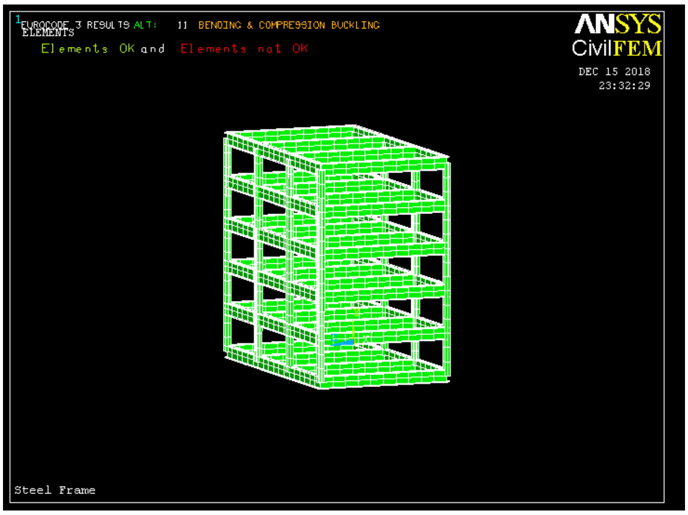

Figure 3.43: Lateral + Compression Buckling Checking Results

The results of checking of combination of lateral and compression buckling by using Eurocode 3 in the Figure 3.43. All checking of the whole steel frame is passed since the green colour represented that elements are ok.

CHAPTER 4

RESULTS AND DATA ANALYSIS

4.1 INTRODUCTION

The 3D steel frame model's results that extracted from the ANSYS software will be further discussed in this chapter. These results include the analysis of graph from the random input and output which obtained from the software. These data are used to analyze hence to form a conclusion. Input and output parameter are the two sources of data. There are two graphs which are probability density and probability in input parameter meanwhile the maximum value of the output parameter in output parameter. There are six input parameters which are, vertical load, wind load, elastic modulus, poison ratio, temperature and density in this 3D steel frame model.

Statistics of the Probabilistic Results, Sample History Plots, Histogram Plots, Cumulative Distribution Function Plots and Sensitivity Plots were included in probabilistic analysis result that obtained from the ANSYS software. In this analysis, total 45 number of samples and 10000 numbers of simulations were used 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

The input variables of this probabilistic analysis to determine the effect of the changing of the variables to the model structure are shown in the below Table 4.1. The symbol of input used in this analysis are named DENS, ELASTIC, POISON, LOAD, WINDLOAD and TEMP.

TABLE 4.1: Random Input Variables Specifications

No.	Name	Type	Par1	Par2
1	DENS	GAUS	7850	1000
2	ELASTIC	GAUS	2.5E+11	1.0E+11
3	POISON	GAUS	0.40	0.10
4	LOAD	GAUS	-20000	10000
5	WINDLOAD	GAUS	3000	1000
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. The density is 5000 kg/m^3 when the probability design is zero according to the Figure 4.1 as shown below. Whereas, the density is around 7600 kg/m^3 when the maximum point of probability density graph is showing 0.0004. 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.

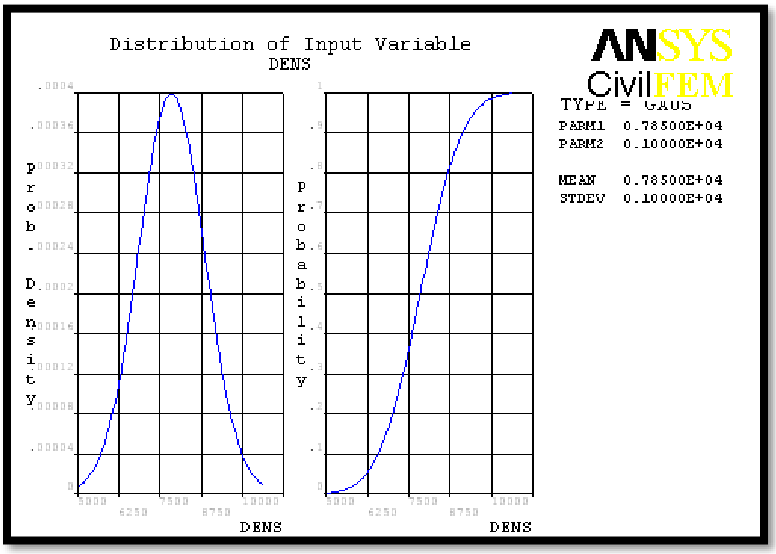

Figure 4.1: PDF & CDF of Input Random Variable DENS

In the probability density graph, the elastic modules is 70GPa when the probability design is zero according to the Figure 4.2 as shown below. Whereas, elastic modulus is around 270GPa when the maximum point of probability density graph is showing 0.00008. 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.

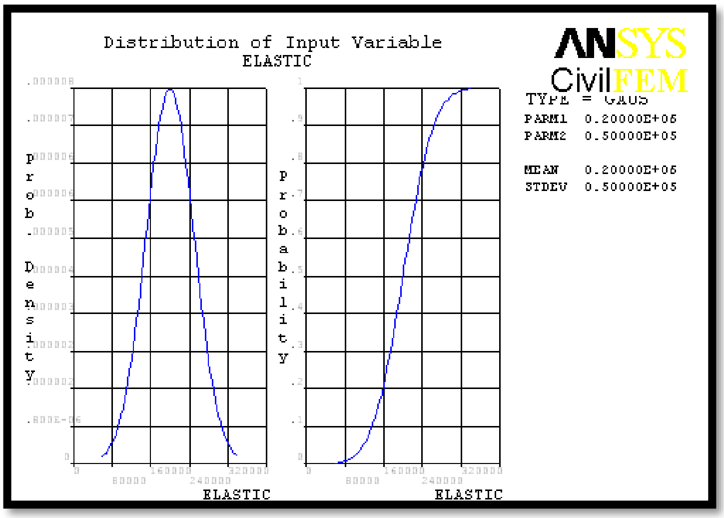

Figure 4.2: PDF & CDF of Input Random Variable ELASTIC

In the probability density graph, the poison ratio is -0.4 when the probability design is zero according to the Figure 4.3 as shown below. Whereas, poison ratio is around 0.2 when the maximum point of probability density graph is showing 1.6. 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.

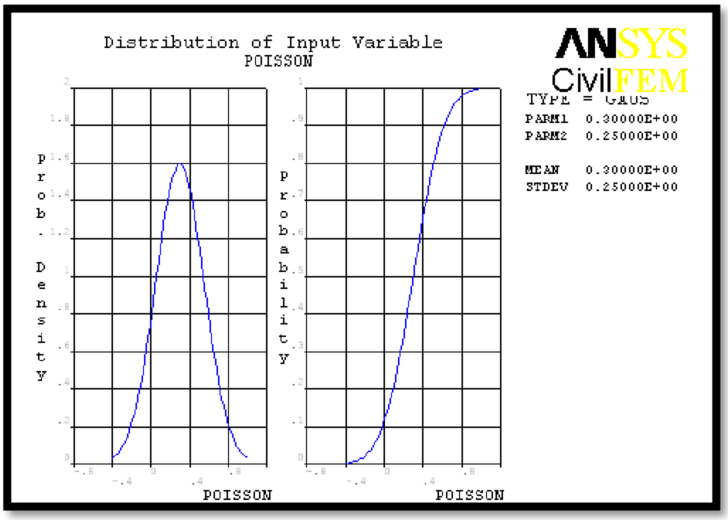

Figure 4.3: PDF & CDF of Input Random Variable POISON

In the probability density graph, the vertical load is around -51kN when the probability design is zero according to the Figure 4.4 as shown below. Whereas, vertical load is around -50kN when the maximum point of probability density graph is showing 0.0008. 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.

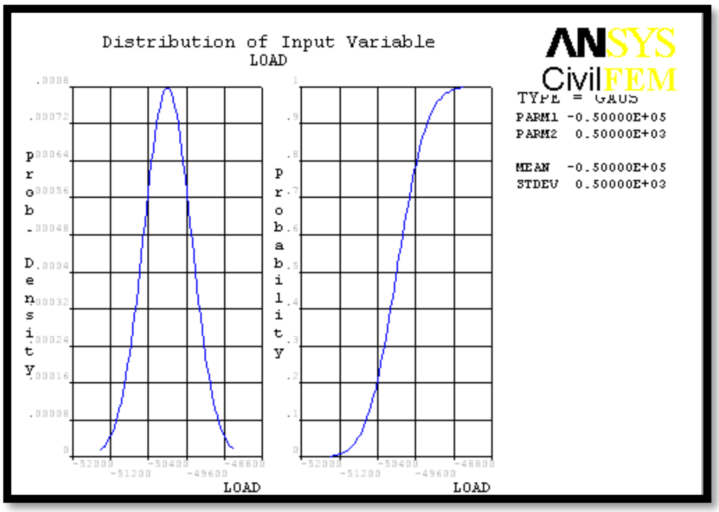

Figure 4.4: PDF & CDF of Input Random Variable LOAD

In the probability density graph, the wind load is around 13kN when the probability design is zero according to the Figure 4.5 as shown below. Whereas, wind load is around 15kN when the maximum point of probability density graph is showing 0.0008. 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.

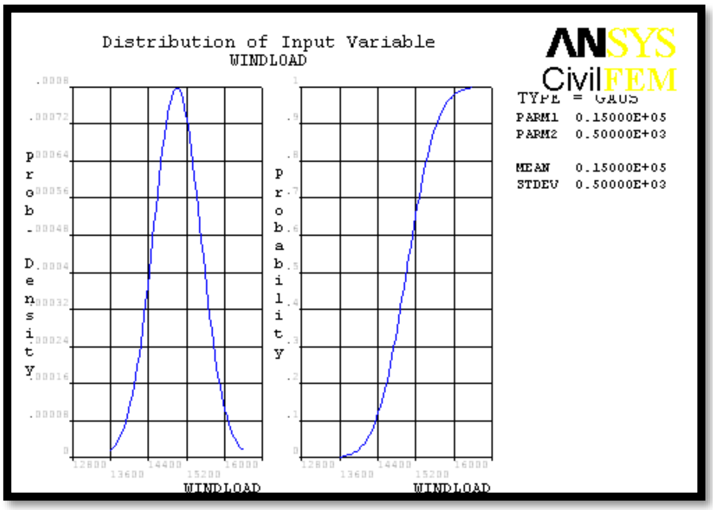

Figure 4.5: PDF & CDF of Input Random Variable WINDLOAD

In the probability density graph, the temperature is around -50ºC when the probability design is zero according to the Figure 4.6 as shown below. Whereas, temperature is around 70ºC when the maximum point of probability density graph is showing 0.01. 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.

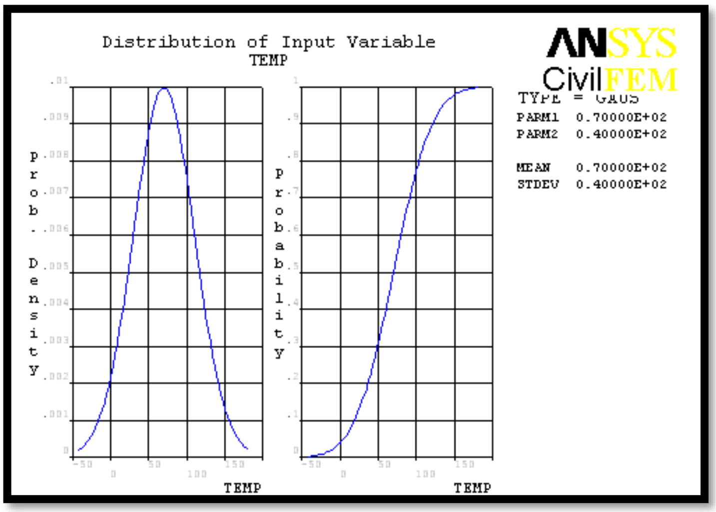

Figure 4.6: PDF & CDF of Input Random Variable TEMP

4.2.2 Statistics of The Probabilistic Results

The statistical analysis of the input parameter is shown in Table 4.2 meanwhile output parameter is shown in Table 4.3. 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 whereas coefficient of skewness is 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 peaked-ness of the probability distribution of a real-valued random variable. 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. 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 value is 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 indicate 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.

The simulation sample for output parameter MAXIMUMDEFLECTION is shown in Figure 4.7. The simulation is running for 30 samples hence each sample has different MAXIMUMDEFLECTION which form the graph as shown.

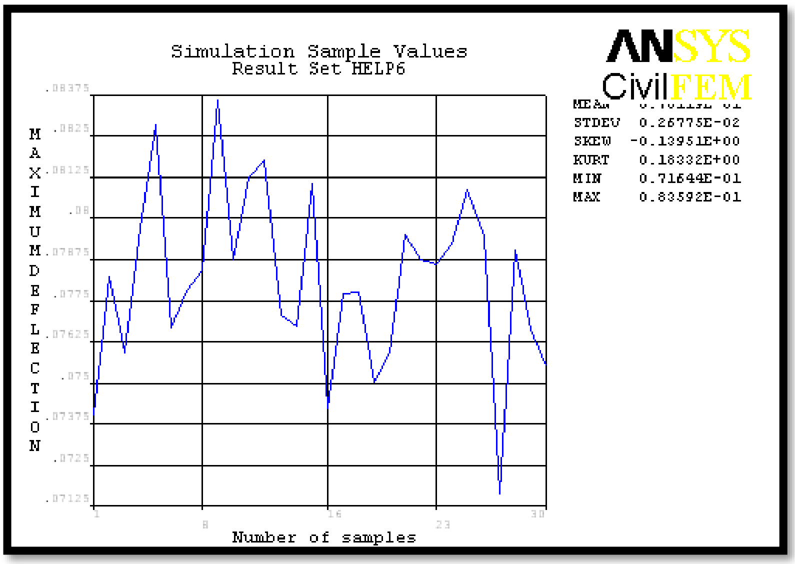

Figure 4.7: Sampled Values for Output Parameter MAXIMUMDEFLECTION

The mean value history for output parameter of MAXIMUMDEFLECTION is shown in Figure 4.8. The mean value of the output parameter is 0.010119 is shown in the graph below. The blue line indicates the mean value meanwhile red line indicates the limit of the mean value for the output parameter by using confidence limit of 95%.

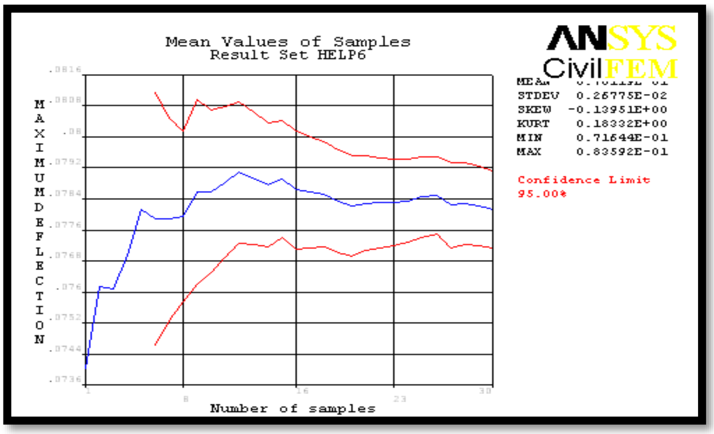

Figure 4.8: Mean Value History for Output Parameter MAXIMUMDEFLECTION

The standard deviation history for output parameter of MAXIMUMDEFLECTION is shown in Figure 4.9. The standard deviation of the output parameter is 0.0026775 is shown in the graph below. The blue line indicates the standard deviation meanwhile red line indicates the limit of the standard deviation for the output parameter by using confidence limit of 95%.

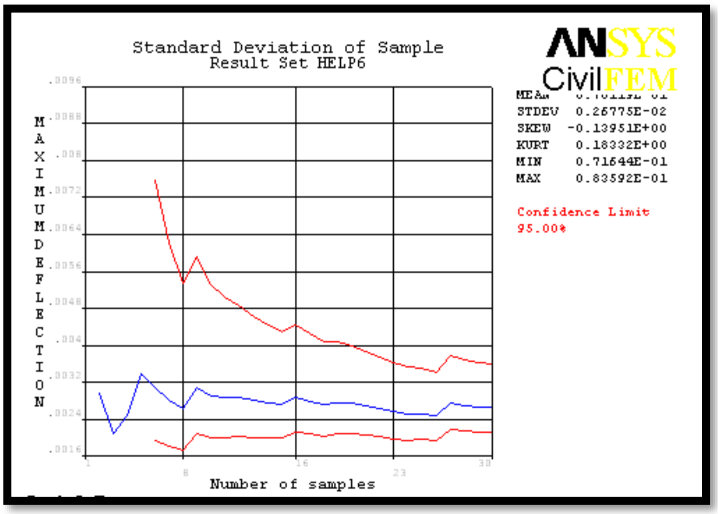

Figure 4.9: Standard Deviation History for Output Parameter MAXIMUMDEFLECTION

The minimum value for output parameter of MAXIMUMDEFLECTION is shown in Figure 4.10. The minimum value of the output parameter is 0.071644 is shown in the graph below. The blue line indicates the minimum value meanwhile red line indicates the limit of the minimum value for the output parameter by using confidence limit of 95%.

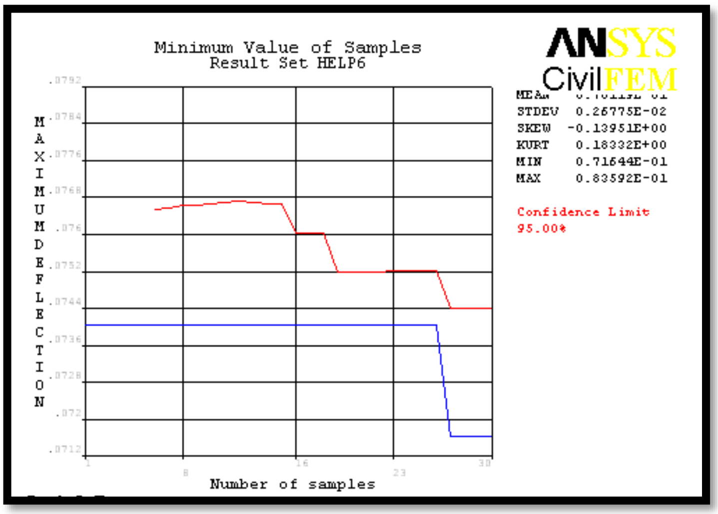

Figure 4.10: Minimum Value for Output Parameter MAXIMUMDEFLECTION

The maximum value for output parameter of MAXIMUMDEFLECTION in Figure 4.11. The maximum value of the output parameter is 0.08259 is shown in the graph below. The blue line indicates the maximum value while red line indicates the limit of the maximum value for the output parameter by using confidence limit of 95%.

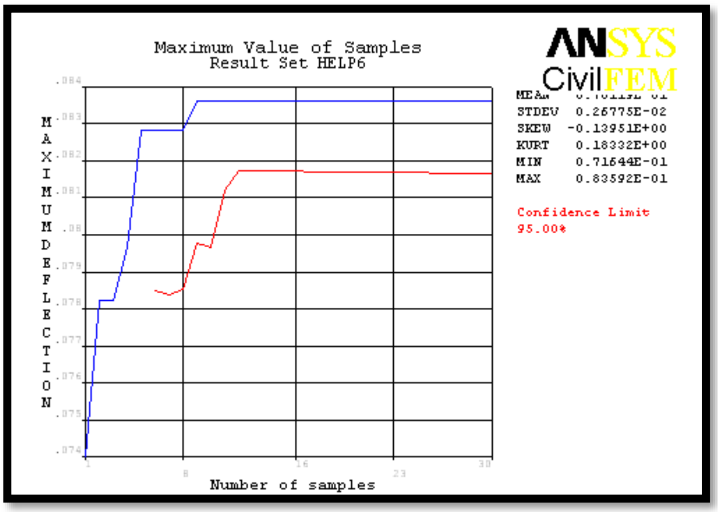

Figure 4.11: Maximum Value for Output Parameter of MAXIMUMDEFLECTION

The simulation sample for output parameter MAX_DEFLECTION is shown in Figure 4.12. 30 samples ran in the simulation whereas each sample has different MAXIMUMDEFLECTION.

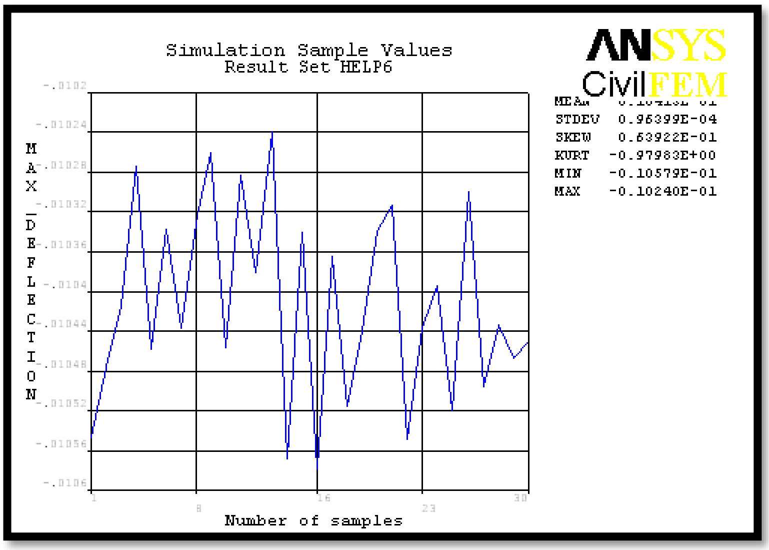

Figure 4.12: Sample Values for Output Parameter MAX_DEFLECTION

The mean value history for output parameter of MAXIMUMDEFLECTION is shown in Figure 4.13. The mean value of the output parameter is 0.010418 is shown in the graph below. The blue line indicates the mean value meanwhile red line indicates the limit of the mean value for the output parameter by using confidence limit of 95%.

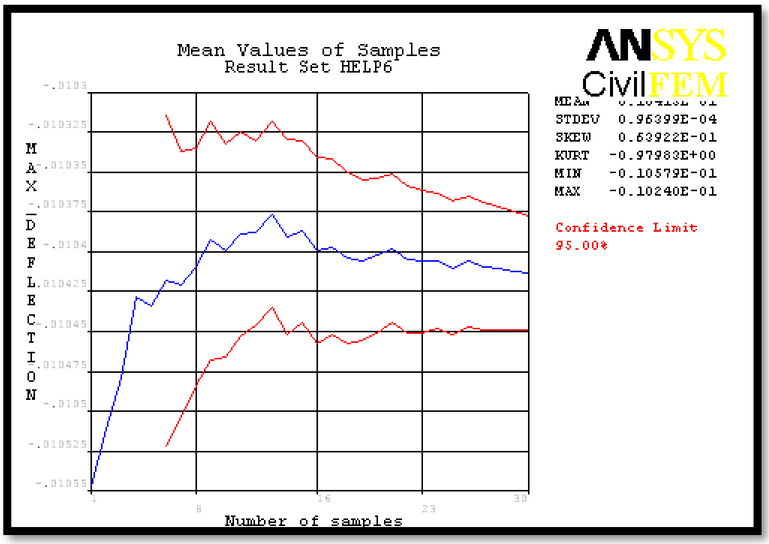

Figure 4.13: Mean Value History for Output Parameter MAX_DEFLECTION

The standard deviation history for output parameter of MAX_DEFLECTION in the Figure 4.14. The standard deviation of the output parameter is 0.00096 is shown in the graph below. The blue line indicates the standard deviation meanwhile red line indicates the limit of the standard deviation for the output parameter by using confidence limit of 95%.

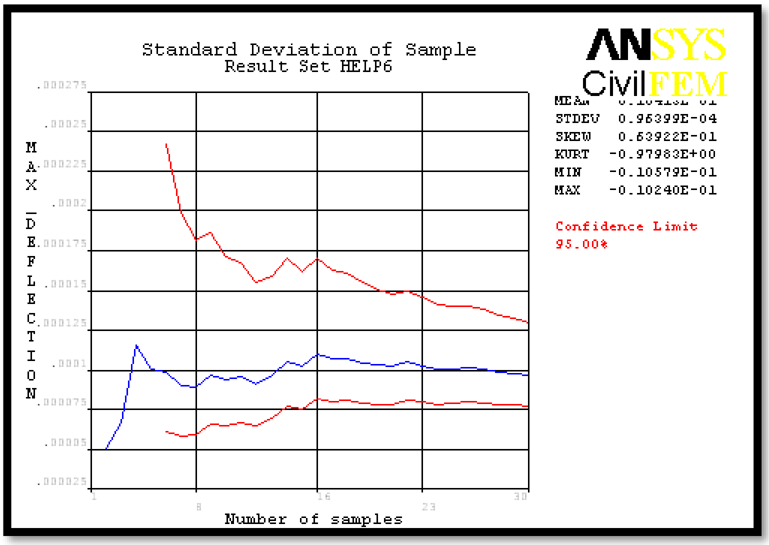

Figure 4.14: Standard Deviation History for Output Parameter MAX_DEFLECTION

The minimum value for output parameter of MAX_DEFLECTION is shown in Figure 4.15. The minimum value of the output parameter is -0.010579 is shown in the graph below. The blue line indicates the minimum value meanwhile red line indicates the limit of the minimum value for the output parameter by using confidence limit of 95%.

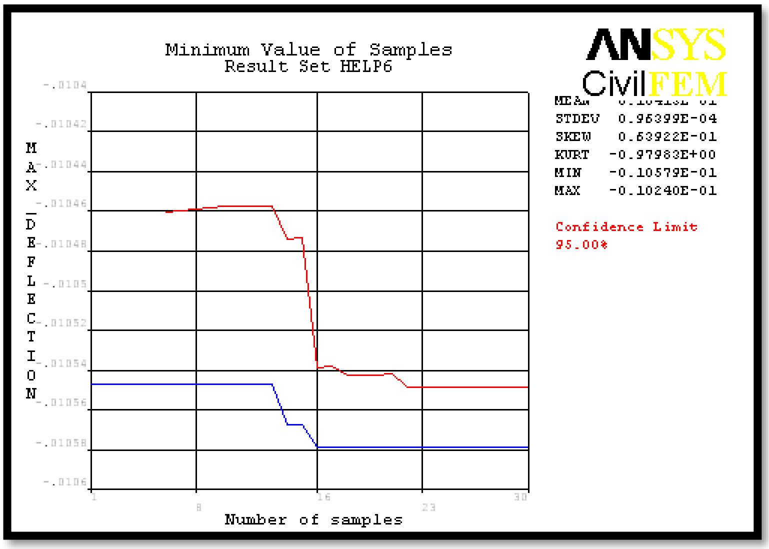

Figure 4.15: Minimum Value for Output Parameter MAX_DEFLECTION

The maximum value for output parameter of MAX_DEFLECTION is shown in Figure 4.16. The maximum value of the output parameter is -0.01024 is shown in the graph below. The blue line indicates the maximum value meanwhile red line indicates the limit of the maximum value for the output parameter by using confidence limit of 95%.

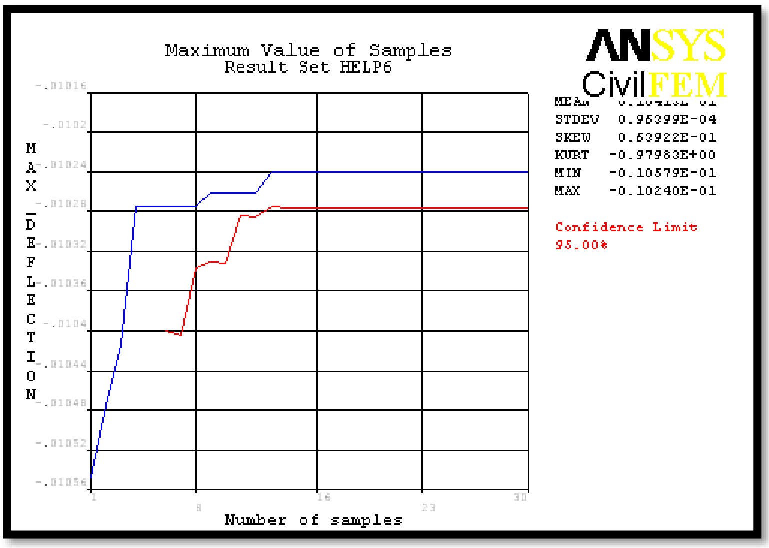

Figure 4.16: Maximum Value for Output Parameter of MAX_DEFLECTION

The simulation sample values for output parameter MAXIMUMDEFLECTION by 10000 numbers of sample is shown in Figure 4.17. The X-axis represents the number of runs in the analysis whereas Y axis represents maximum defection. The mean is around 0.01m according to the simulation results. In other hands, the minimum is 0.063251m and maximum is 0.089028m.

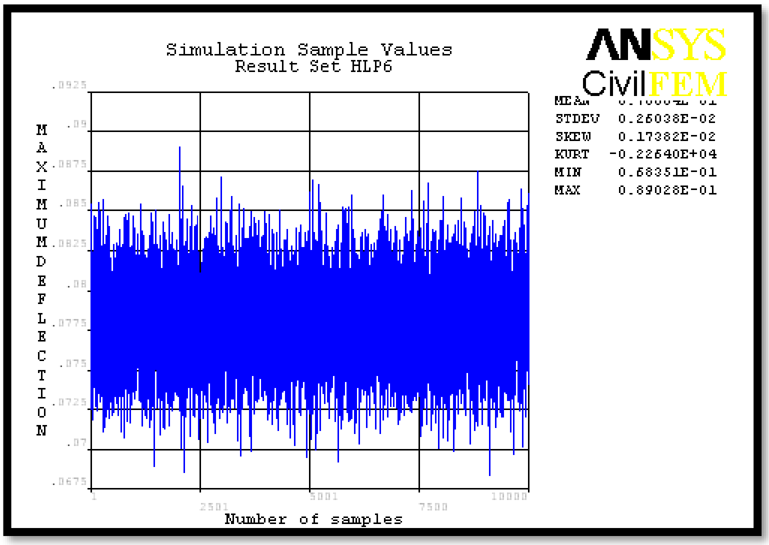

Figure 4.17: Simulation Sample Values for Output Parameter MAXIMUMDEFLECTION

The simulation sample values for output parameter MAX_DEFLECTION is shown in Figure 4.18. The X-axis represents the number of runs in the analysis whereas Y axis represents maximum defection. The mean value is 0.01m according to the simulation results. Besides, the value for maximum is -0.099576m and minimum value is -0.010807m.

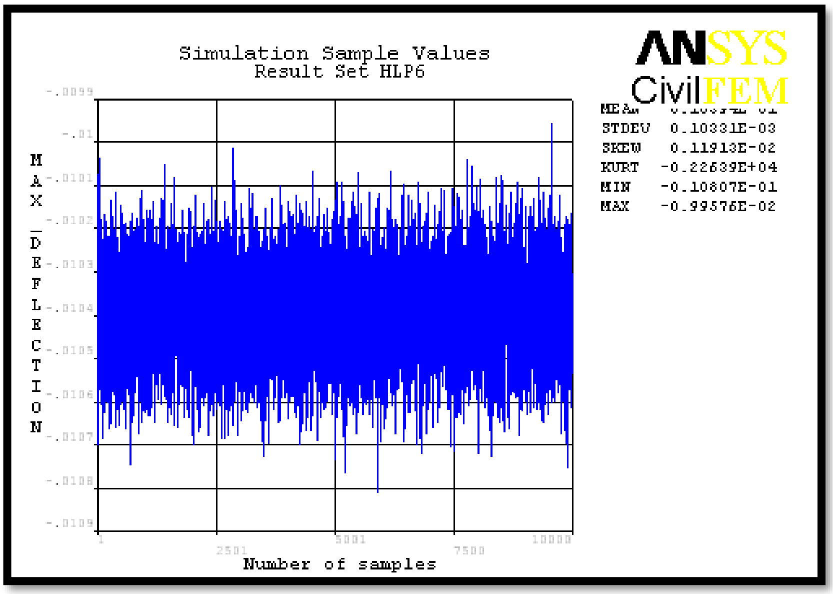

Figure 4.18: Simulation Sample Values for Output Parameter MAX_DEFLECTION

4.2.4 Histogram Plots of Output Parameters

An appropriate number of classes based on the number of samples is calculated by ANSYS PDS. The number of bars is equal to the number of classes that shown in the histogram. Same width obtained from the divided range between the smallest and largest sample value is given to every class. A histogram is obtained by the number in each individual class and the number is divided by the total number of samples. Therefore, a histogram is a graphical representative of the distribution data. The histogram of output parameter for MAXIMUMDEFLECTION is shown in Figure 4.19 whereas the histogram of output parameter for MAX_DEFLECTION is shown in Figure 4.20.

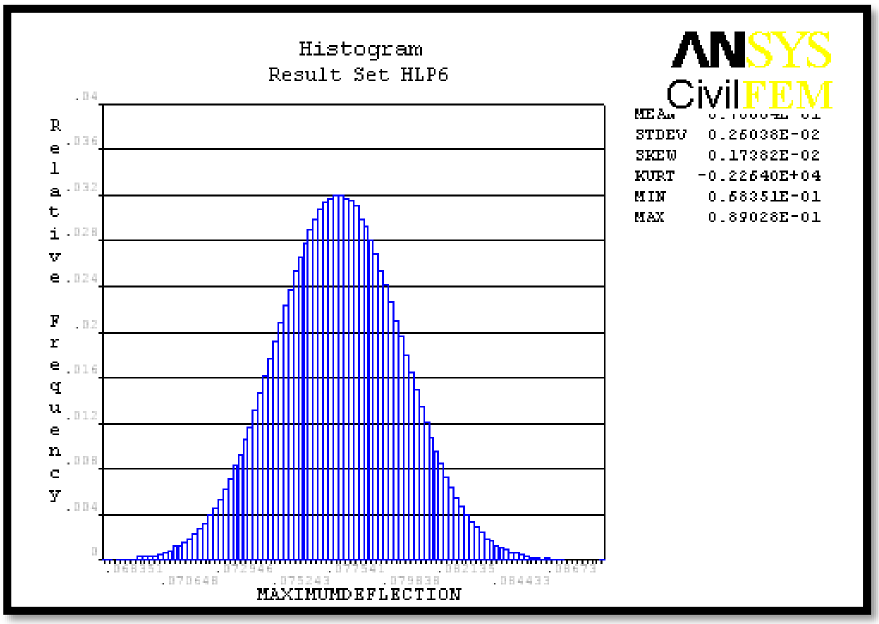

Figure 4.19: Histogram of Output Parameter for MAXIMUMDEFLECTION

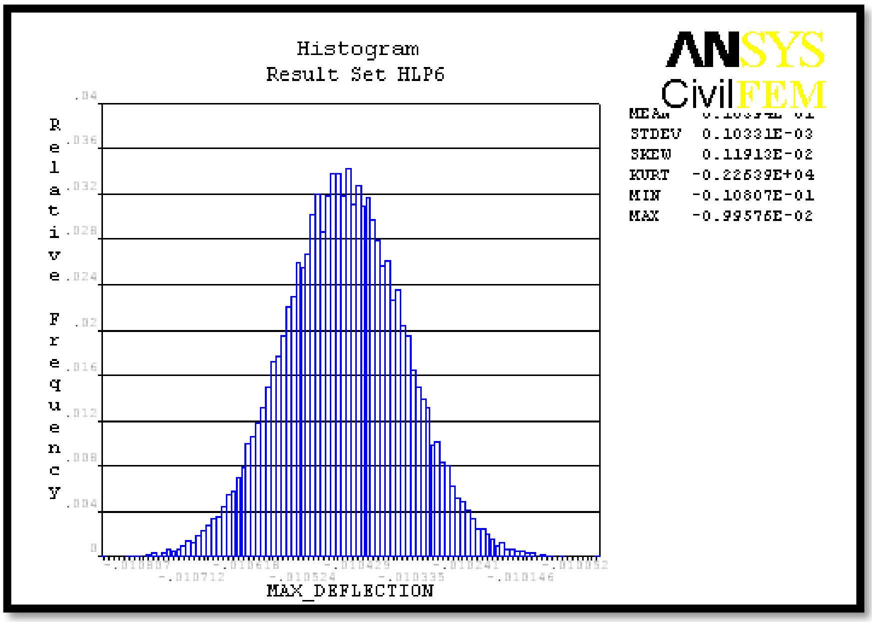

Figure 4.20: Histogram of Output Parameter for MAX_DEFLECTION

The technical statistics information on the coefficient of skewness and coefficient of kurtosis is shown in Figure 4.19 and 4.20. In coefficient of skewness, 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. The values of skewness are 0.0017382 and 0.0011913 that shown in Figure 4.19 and 4.20. Since these values are very close to zero hence they are acceptable.

Coefficient of kurtosis, a measure of the peaked-ness of the probability distribution of a real-valued random variable, although some sources are insistent that heavy tails, and not peaked-ness, 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. The kurtosis statistic of -0.0002264 and -0.00022639 that shown in Figure 4.19 and 4.20. Since the values are very close to zero then they are acceptable.

Besides, the minimum and maximum deflection of the model is 68.351mm and 89.028mm respectively based on Figure 4.19. Whereas, the minimum and maximum deflection of the model is -10.807mm and -9.9576mm respectively based on Figure 4.20.

4.2.5 Histogram Plots of Input Parameters

The value of skewness is -0.00075805 is shown in the Figure 4.21. Since the value is very close to zero hence it is acceptable. We can conclude that it is normal distribution set of test scores whereas it is probably just the chances fluctuation from zero. The tail on the left side is longer than right side. The histogram has kurtosis statistic of -2261.7 as it has infrequent extreme deviations. Besides, the minimum and maximum density of the model is 3814kg/m^3 and 11710kg/m^3 respectively.

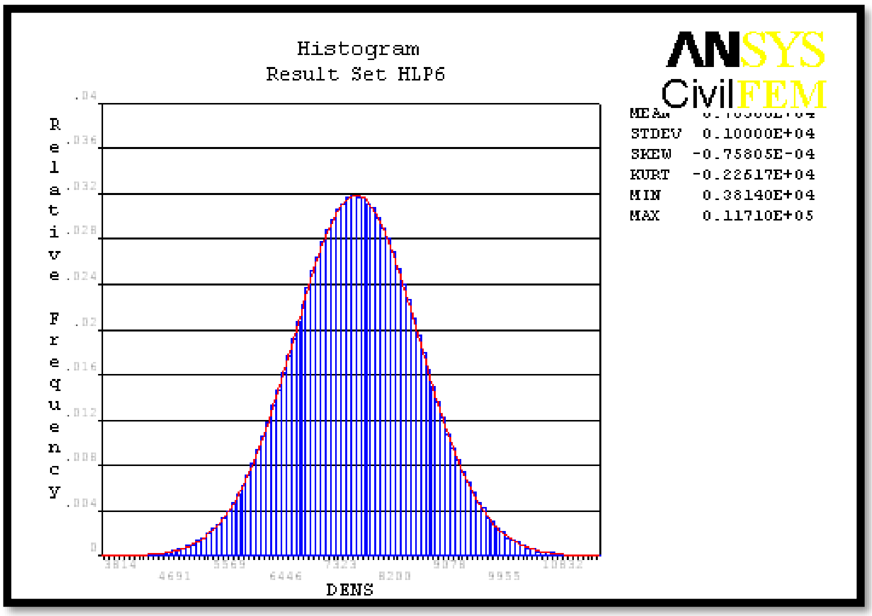

Figure 4.21: Histogram of Input Parameter for DENS

The value of skewness is 0.00201 is shown in Figure 4.22. Since the value is very close to zero hence it is acceptable. We can conclude that it is normal distribution set of test scores and it is probably just the chances fluctuation from zero. The tail on the right side is longer than left side. The histogram has kurtosis statistic of -2263.4 as it has infrequent extreme deviations. Besides, the minimum and maximum elastic modulus of the model is -174.91GPa and 615.51GPa respectively.

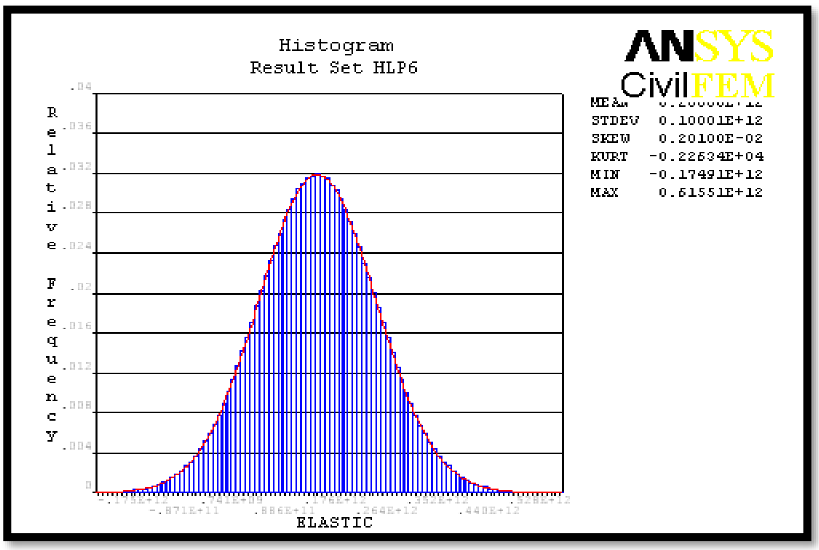

Figure 4.22: Histogram of Input Parameter for ELASTIC

The value of skewness is -0.0010672 is shown in Figure 4.23. Since the value is very close to zero hence it is acceptable. We can conclude that it is normal distribution set of test scores and it is probably just the chances fluctuation from zero. The tail on the left side is longer than right side. The histogram has kurtosis statistic of -2262.8 as it has infrequent extreme deviations. Besides, the minimum and maximum poison ratio of the model is -0.72816 and 1.2521 respectively.

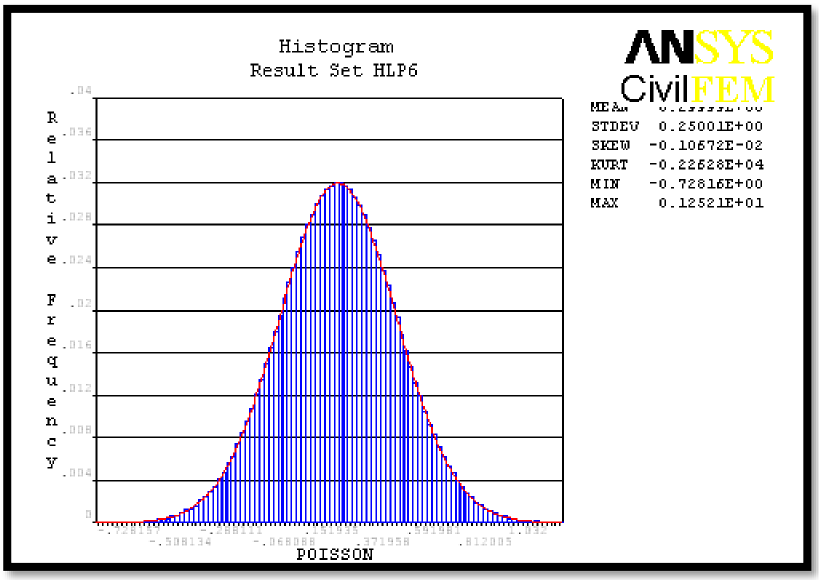

Figure 4.23: Histogram of Input Parameter for POISON

The value of skewness is 0.00164 is shown in Figure 4. Since the value is very close to zero hence it is acceptable. We can conclude that it is 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. The histogram has kurtosis statistic of -2255.5 as it has infrequent extreme deviations. Besides, the minimum and maximum load of the model is -52kN and -47.89kN respectively.

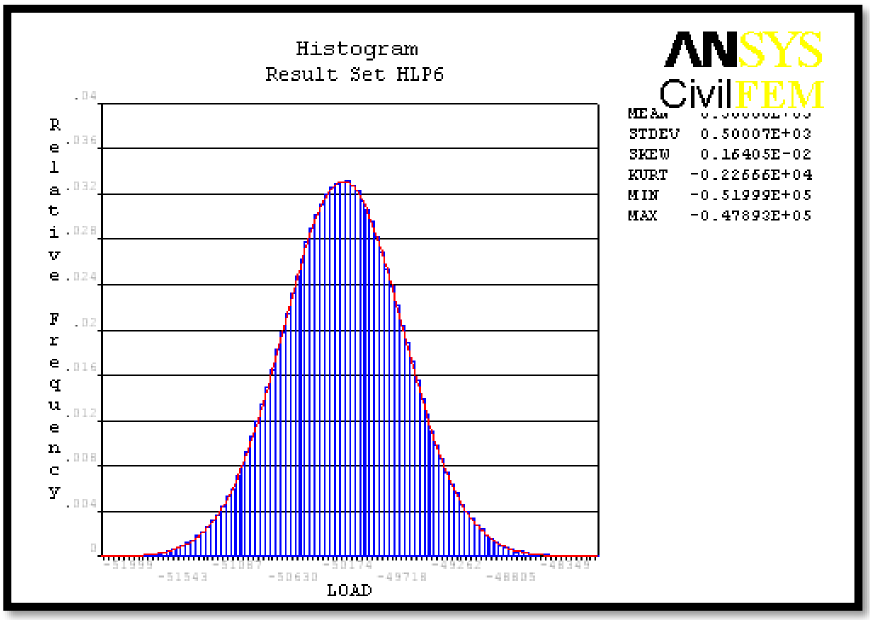

Figure 4.24: Histogram of Input Parameter for LOAD

The value of skewness is 0.0017444 is shown in the Figure 4.25. Since the value is very close to zero hence it is acceptable. We can conclude that it is normal distribution set of test scores and it is probably just the chances fluctuation from zero. The tail on the right side is longer than left side. The histogram has kurtosis statistic of -2264 as it has infrequent extreme deviations. Besides, the minimum and maximum wind load of the model is 13131kN and 17102kN respectively.

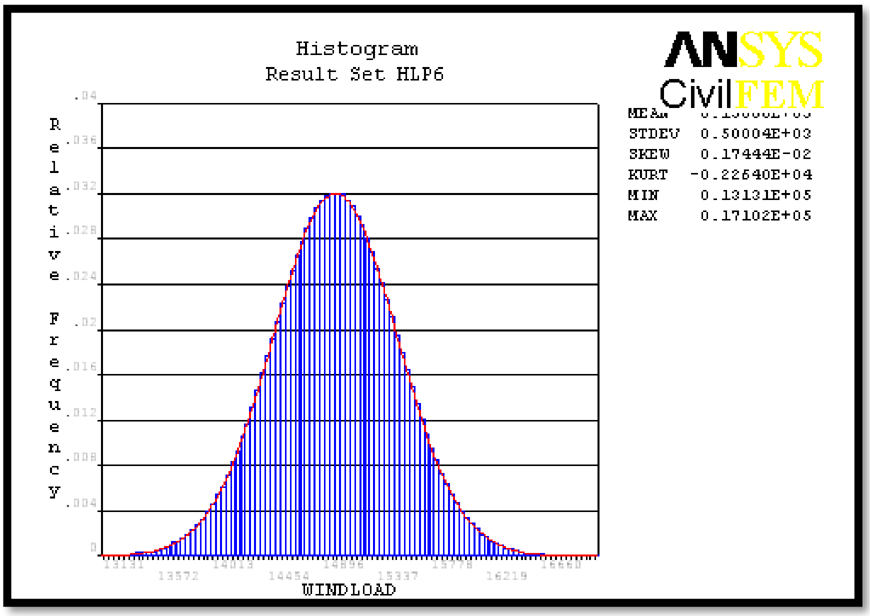

Figure 4.25: Histogram of Input Parameter for WINDLOAD

The value of skewness is 0.0051523 is shown in the Figure 4.26. Since the value is very close to zero hence it is acceptable. We can conclude that it is normal distribution set of test scores and it is probably just the chances fluctuation from zero. The tail on the right side is longer than left side. The histogram has kurtosis statistic of -2278.5 as it has infrequent extreme deviations. Besides, the minimum and maximum temperature of the model is -82.196ºC and 262.64ºC respectively.

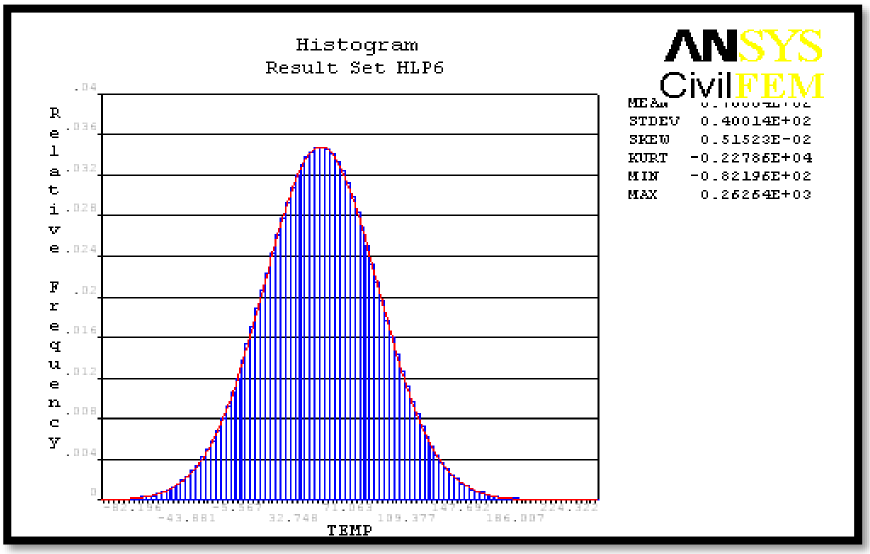

Figure 4.26: Histogram of Input Parameter for TEMP

4.2.6 Cumulative Distribution Function Plots

The summation of relative frequency from zero percent to hundred percent is used to plot the cumulative distribution curve. The upper and lower curves are the confidence interval using a 95% confidence level according to Figure 4.27 to Figure 4.34 as shown below. The accuracy of the probability results is based on the confidence interval. The resulting value may not sufficient after the reliability of the steel frame has been quantified. So, probabilistic method is used by inputting variables. Hence, the results can be obtained from the probabilistic sensitivity diagrams plot.

All the deflection value should be falling between the upper and lower curves for the stability of the steel frame.

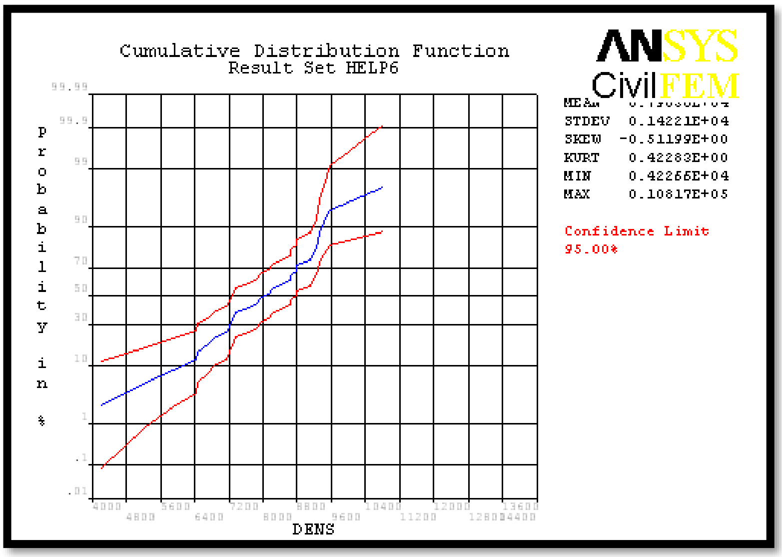

Figure 4.27: CDF of Input Variable DENS

The maximum value of variable DENS is 10817kg/m^3 meanwhile the minimum value is 4226.6kg/m^{3}as shown in Figure 3.27. The mean value of density is falling between the maximum and minimum value which is 7850kg/m^3. This indicates that the value of density is allowed to fluctuate between the range of two curves.

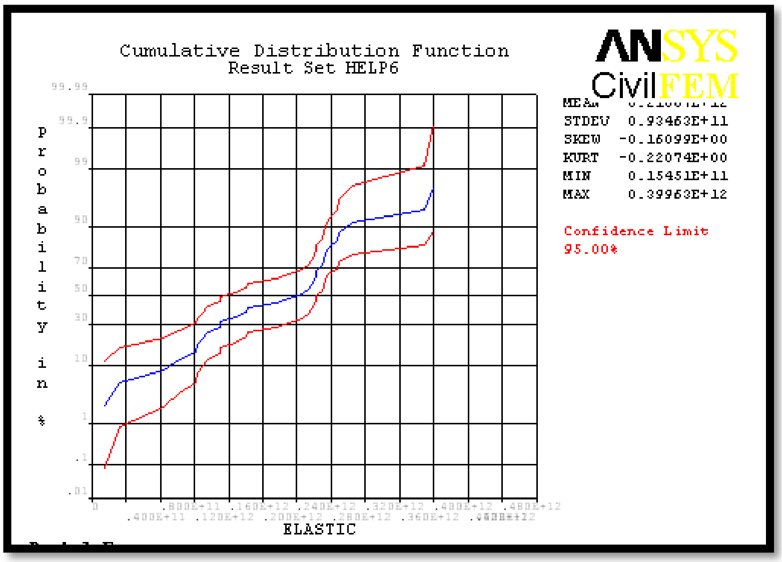

Figure 4.28: CDF of Input Variable ELASTIC

The maximum value of variable ELASTIC is 399.63GPa meanwhile the minimum value is 15.451GPa as shown in Figure 4.28. The mean value of elastic modules is falling between the maximum and minimum value which is 210GPa. This indicates that the value of elastic modulus is allowed to fluctuate between the range of two curves.

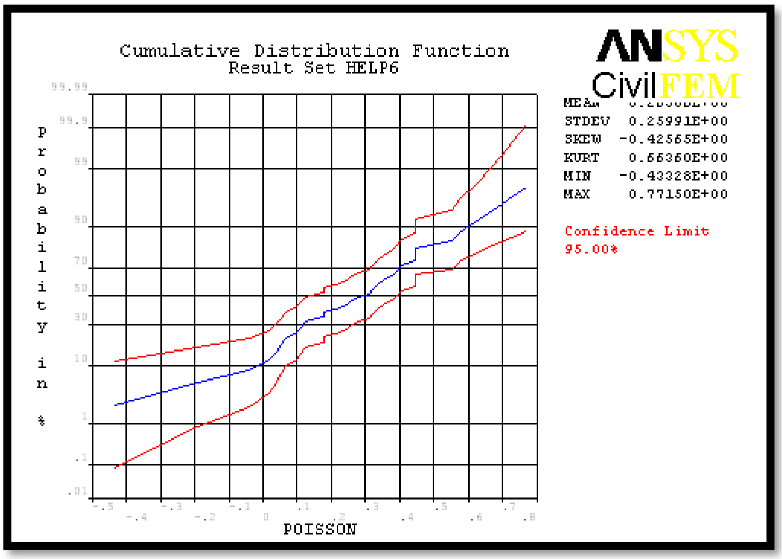

Figure 4.29: CDF of Input Variable POISON

The maximum value of variable POISON is 0.7715 meanwhile the minimum value is -0.43328 as shown in Figure 4.29. The mean value of poison ratio is falling between the maximum and minimum value which is 0.2. This indicates that the value of poison ratio is allowed to fluctuate between the range of two curves.

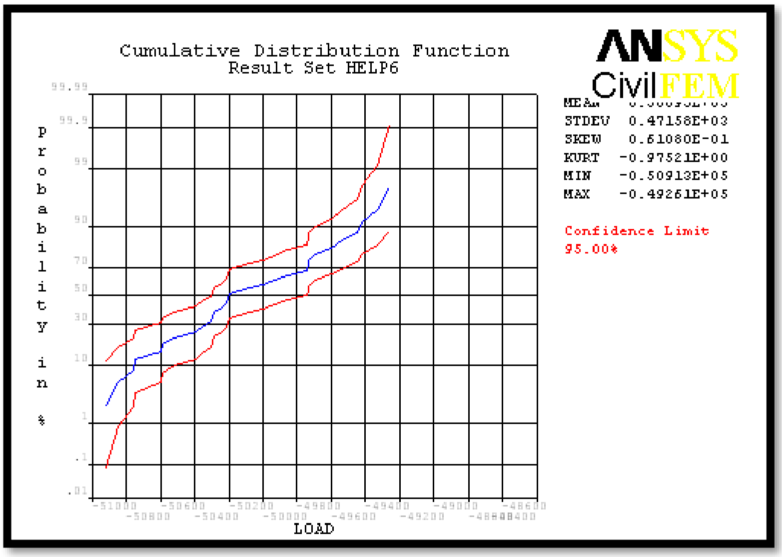

Figure 4.30: CDF of Input Variable LOAD

The maximum value of variable LOAD is -49.261kN meanwhile the minimum value is -50.913kN as shown in Figure 4.30. The mean value of load is falling between the maximum and minimum value which is 50kN. This indicates that the value of load is allowed to fluctuate between the range of two curves

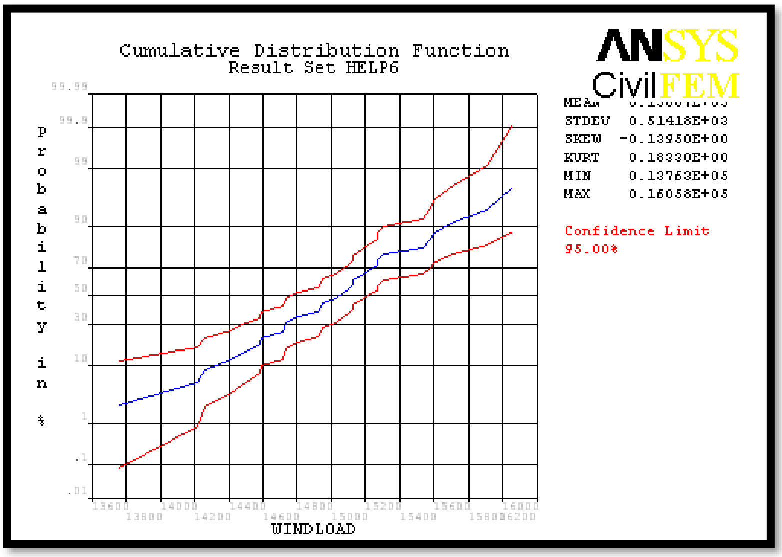

Figure 4.31: CDF of Input Variable WINDLOAD

The maximum value of variable WINDLOAD is 16.058kN meanwhile the minimum value is 13.763kN as shown in Figure 4.31. The mean value of wind load is falling between the maximum and minimum value which is 15kN. This indicates the value of wind load is allowed to fluctuate between the range of two curves.

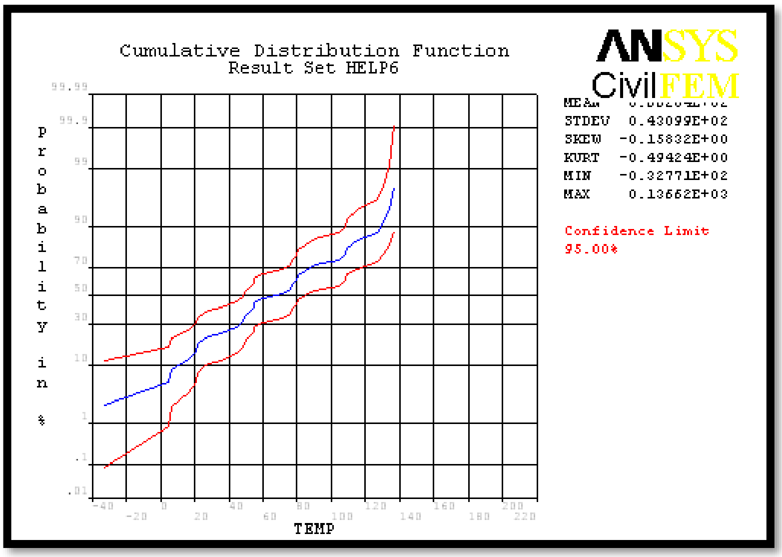

Figure 4.32: CDF of Input Variable TEMP

According to Figure 4.32 as shown above, the maximum value of variable TEMP is 136.62ºC meanwhile the minimum value is -32.77ºC. The mean value of temperature is falling between the maximum and minimum value which is 70ºC. This indicates the value of temperature is allowed to fluctuate between the range of two curves.

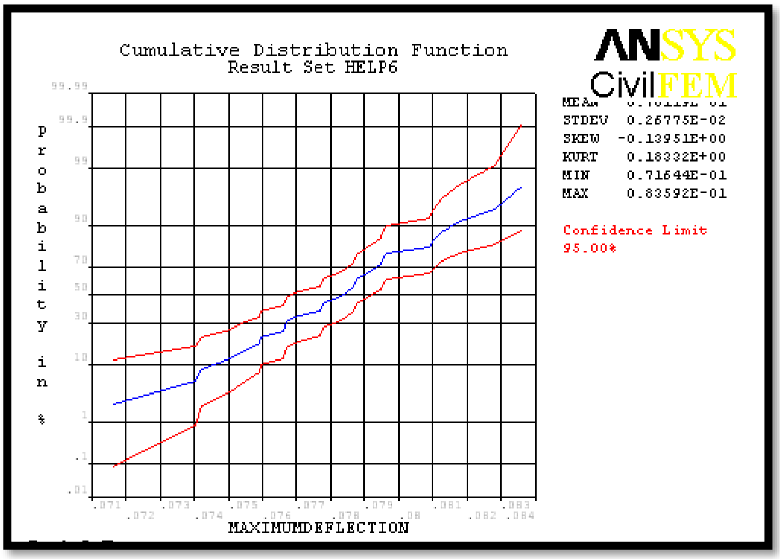

Figure 4.33: CDF of Output Variable MAXIMUMDEFLECTION

The maximum value of variable MAXIMUMDEFLECTION is 83.592mm meanwhile the minimum value is 71.644mm as shown in Figure 4.33. This indicates the value of maximum deflection is allowed to fluctuate between the range of two curves.

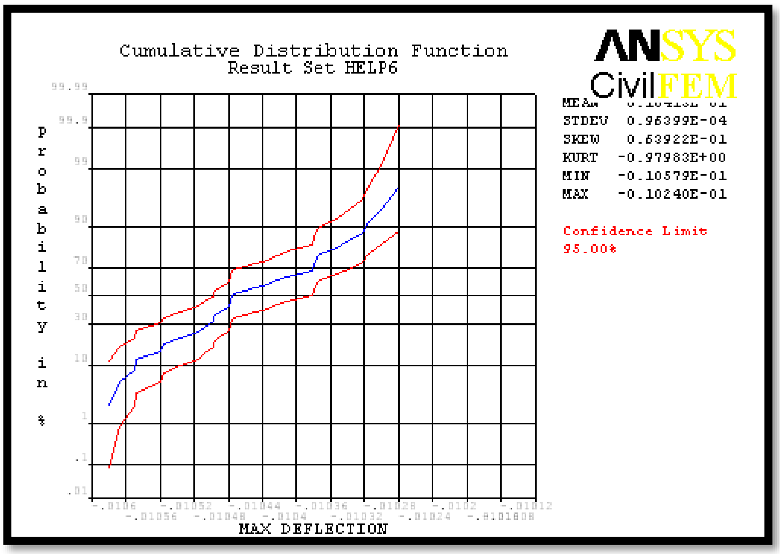

Figure 4.34: CDF of Output Variable MAX_DEFLECTION

The maximum value of variable MAX_DEFLECTION is -10.24mm meanwhile the minimum is -10.579mm as shown in Figure 4.34. This indicates 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. The significant inputs are considered whereas the insignificant inputs are ignored to plot the sensitivities pie chart. Significant input defined as the output values is affected highly by input value when compared to the insignificant inputs.

The effect of significant random input to the MAXIMUMDEFLECTION is shown in Figure 4.35. According to Figure 4.35, two variables which are input WINDLOAD and LOAD affect the deflection of the steel frame. Meanwhile, the other variables are insignificant. Despite that, WINDLOAD has the highest sensitivity to the deflection of steel frame. WINDLOAD variable is 55% significant and LOAD variable is 45% significant to the output MAXIMUMDEFLECTION whereas the other variables are insignificant through the interpretation from the figure below.

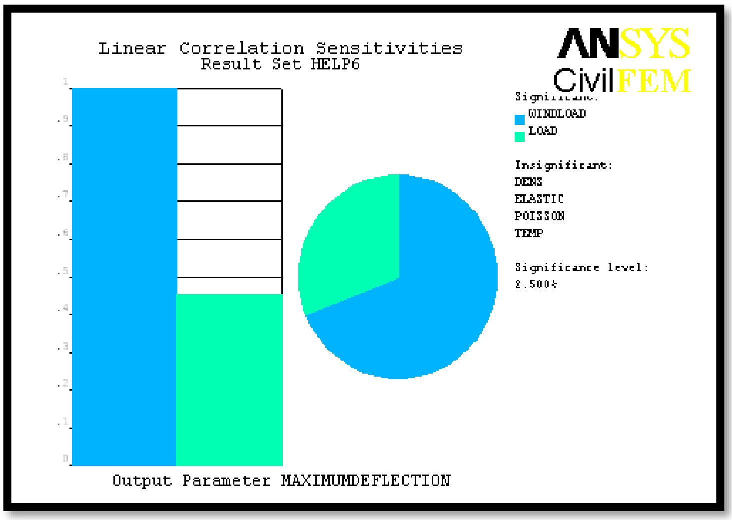

Figure 4.35: Sensitivity Plot for MAXIMUMDEFLECTION

The effect of significant random input to the MAX_DEFLECTION as shown in Figure 4.36. According to Figure 4.35, three variables which are input ELASTIC, WINDLOAD and LOAD affect the deflection of the steel frame. Meanwhile, the other variables are insignificant. Despite that, LOAD has the highest sensitivity to the deflection of steel frame. ELASTIC variable is also 4% significant, WINDLOAD variable is 4% significant and LOAD variable is 92% significant to the output MAX_EFLECTION whereas the other variables are insignificant through the interpretation from the figure below.

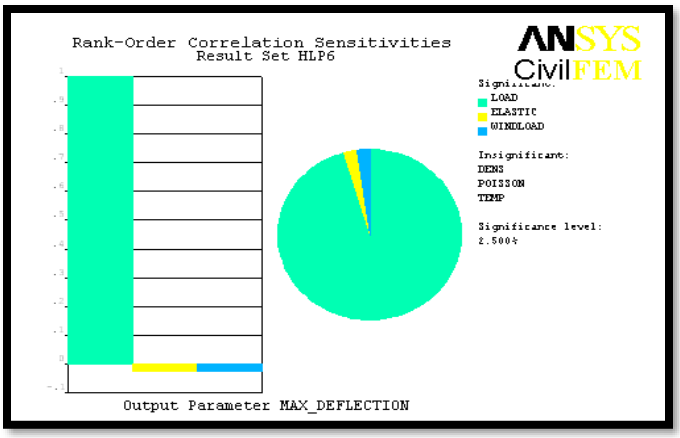

Figure 4.36: Sensitivity Plot for MAX_DEFLECTION

4.2.8 Linear Correlation Coefficients

The linear correlation coefficients between input variables, between input and output variables and between output variables are shown in Table 4.4, 4.5 and 4.6. The value indicates the degree of linear dependence between the variables. The positive value indicates the linear relationship is increasing which means perfect direct relationship. Other than that, negative value indicates the linear relationship is decreasing which means anticorrelation. When the value is 1, it means that the correlation between the variables, vice versa. From the tables below, values of variables of DENS/DENS, ELASTIC/ELASTIC and so on are 1 since they are linearly dependence. POISON/WINDLOAD indicates the lowest positive value of 0.026 which means least increase linear relationship whereas TEMP/LOAD and TEMP/ELASTIC are 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
MAXIMUMD EFLECTION	0.056	0.044	0.026	0.451	1.000	0.163
MAX_DEFLE CTION	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). The spearman rank order correlation coefficients between input variables, between input and output variables and between output variables are shown in Table 4.7, 4.8 and 4.9. There are two types of sign in the tables below which are negative and positive. Positive value indicates the Y tends to increase when X increases whereas negative value indicates the Y tends to decrease when X increases. Spearman correlation of +1 or -1 are indicating that the correlation is perfect. 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 according to the tables below.

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 research, has demonstrated a method of probabilistic and sensitivity analysis to assess the steel frame is proposed. Direct sampling simulation technique was used to study the uncertain of model parameters. The sample are obtained according to underlying probabilistic distribution and outputs from the numerical simulation are translated into probabilistic distribution. Probabilistic methods provide a computationally efficient method for treating uncertainties in engineering calculation.

5.2 CONCLUSION

The analysis result of steel frame was similar between ANSYS result and manual calculation. Besides, structural of the steel frame also passed in term of shear force, bending moment and so on according to the ANSYS results. 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, Wind Load, Temperature, Density, Elastic and Poison. Through the study, investigate, examine and establish the effect methods to predict the main parameter uncertainty on the final results are accomplished.

In nutshell, only 3 inputs affect the results and these defined as significant inputs. In term of identifying more significant inputs for this research, it is utmost to carry out the research. This is due to the sample of probabilistic study on 3D steel frame is very limited. Therefore, more research with different significant input should be carried out.

5.3 RECOMMENDATION

This research is done accordingly to the requirement. Despite that, some improvements need to be done in order to increase the accuracy of this results. There are some recommendations are proposed to improve and gather more information thus enhance the whole research. The recommendations are as shown below:

(i) More significant inputs to be provided
(ii) More details 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.*

BS EN 1993-3-8:. (2005). *Eurode 3: Design of Steel Structures. Design of joints, BSI.*

Caprani, C. (2009/2010). Plastic Analysis Structural Engineering.

Davidson, B., & W.Owens, G. (2012). *Steel Designers' Manual.* BLACKWELL PUBLISHING LTD.

Deakin, U. (2009). Retrieved from Frame Concstruction: http://www.ab.deakin.edu.au/online/vgallery/2004/srt251/team22/Home/navigation/Framing/Frame.htm#top

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.

Goswami, S., Chakraborty, S., & Ghosh, S. (2013). Adaptive Response Surface Method in Structural Approximation Under Uncertainty.

Jing Ji, W. Z. (2012). Research of Seismic Testing and Dynamic Character of High-rise Building Structure Based on ANSYS.

Lenka Lausovaa, I. K. (2017). Numerical Analysis of Steel Frame Exposed to Fire. *Procedia Engineering*.

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

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., & Sha, W. (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: https://www.steelconstruction.info/Continuous_frames

Tvrda, K. (2017). RSM Method in Probabilistic Analysis of the Foundation Plate.

University of Alberta - ANSYS Tutorials. (2010). Retrieved from http://www.mece.ualberta.ca/tutorials/ansys/

APPENDIX A1

EUROCODE 3 CHECKING OF PORTAL FRAME

Structure Properties

Spacing of steel frame = 5m

Spacing between purlins = 1.5m

Member Properties

Cross section: *IPE A 600*

Material: *Fe E355*

Section Properties

h = 597mm	A = 13700mm^2
b = 220mm	I_y = 118 x 10^{4}mm^4
t_w = 9.8mm	I_z = 3116 x 10^{4}mm^4
t_f = 17.5mm	$W_{pl,y}$ = 442.1 x 10^{3}mm^3
r = 24mm	i_z = 43.1mm
h_w = 562mm	I_w = 1249 x 10^{9}mm^6
d = 514mm	

Design of Column

Height of column = 11m

N_{Ed} = 140kN

V_{Ed} = 16kN

M_{Ed} = 17kNm

<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 :

$369\varepsilon/(13\alpha\text{-}1)$ = (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-(2x220x17.5)+[9.2+2(24)]17.5 = 7011.5mm^2

$\eta h_w t_w$ = 1.0x562x9.8 = 5507.6mm^2 < A_v = 7011.5mm^2

$V_{pl,Rd} = [A_v(f_y/\sqrt{3})/\gamma_{M0}]$ = [7011.5(355/$\sqrt{3}$)/1.0]x10^{-3} = 1437.1kN > V_{Ed} = 16kN

Therefore, the shear resistance checking is satisfactory.

Bending & Shear Interaction:

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

Therefore, the Bending & Shear checking is satisfactory.

Compression Resistance:

$N_{c,Rd} = (Af_y)/\gamma_{M0}$ = (13700x355)x10^{-3} = 4863.5kN

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

Therefore, the Compression Resistance checking is satisfactory.

Bending & Axial Force Interaction:

$0.25N_{pl,Rd}$ = 0.25x4863.5 = 1215.88kN

$(0.5h_w t_w f_y)/\gamma_{M0}$ = [(0.5x562x9.8x355)/1.0]x10^{-3} = 977.6kN

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

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

Bending Moment Resistance:

$M_{pl,Rd}$ = $(w_{py}f_y)/\gamma_{M0}$ = [(442.1x10^{-3}x355)/1.0] x10^{-6} = 156.95kNm > M_{Ed} = 17kNm

Therefore, the Bending Moment Resistance checking is satisfactory.

Out-of-Plane Buckling

Verification of spacing between intermediate restraints

C_1 = 1.77

$$L_m = \frac{38 i_z}{\sqrt{\frac{1}{57.4}\left(\frac{N_{Ed}}{A}\right)+\frac{1}{756C_1^2}\frac{W_{pl,y}^2}{AI_t}\left(\frac{f_y}{235}\right)^2}} = \frac{38\times 43.1}{\sqrt{\frac{1}{57.4}\left(\frac{140000}{13700}\right)+\frac{1}{756\times 1.77^2}\frac{442100^2}{13700\times 89300}\left(\frac{355}{235}\right)^2}} = 1698.03\text{mm}$$

Side rail spacing = 2750 > L_m = 1698.03mm

Whole Column (11000mm):

h/b = 597/220 = 2.71

t_f = 17.5mm

curve b

α = 0.34

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

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

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

$$\chi_z = \frac{1}{\text{Ø}z+\sqrt{\text{Ø}_z^2-\lambda_z^2}} = \frac{1}{6.68+\sqrt{6.68^2-3.36^2}} = 0.08$$

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

Therefore, the checking is satisfactory.

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

C_1 = 1.77

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

= 1108.8kNm

$\lambda_{LT} = \sqrt{\frac{W_y f_y}{M_{cr}}} = \sqrt{\frac{442.1\times10^3\times355}{1108.8\times10^6}}$ = 0.38

$\lambda_{LT,0}$ = 0.4

β = 0.75

h/b = 597/220 = 2.71

curve c

$\lambda_{LT} = \frac{L_{cr}}{i_z}\frac{1}{\lambda 1}$ = (11000/43.1)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(0.38-0.4)+(0.75x0.38^2)] = 0.55

$\chi_{LT} = \frac{1}{\emptyset LT+\sqrt{\emptyset_{lT}^2-\beta\lambda_{LT}^2}} = \frac{1}{0.55+\sqrt{0.55^2-0.75\times0.38^2}}$ = 1

$M_{b,Rd} = \frac{\chi_{LT} W_{pl,y} f_y}{\gamma_{M1}} = \frac{1.0\times442.1\times10^3\times355}{1.0}$ x10^{-6} = 156.95kNm > M_{Ed} = 17kNm

Therefore, the checking is satisfactory.

Interaction of Axial Force and Bending Moment:

λ_z = 3.36 > 0.4

ψ=0

C_{mLT} = 0.6+0.4ψ = 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$\left[\left(1-\frac{0.1\times3.36}{0.6-0.25}\frac{140}{389}\right);\left(1-\frac{0.1}{0.6-0.25}\frac{140}{389}\right)\right]$ = max[0.654;0.897] = 0.897

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

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

Therefore, the checking is satisfactory.

In Plane Buckling

h/b = 597/220 = 2.71

t_f = 17.5mm

buckling about y axis

curve a

α = 0.21

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

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

$Ø_y = 0.5[1+\alpha_y(\lambda_y-0.2)\ \lambda_y^2]$ = 0.5[1+0.21(3.36-0.2)+3.36^2] = 6.48

$\chi_y = \frac{1}{Øy+\sqrt{Ø_y^2-\lambda_y^2}} = \frac{1}{6.48+\sqrt{6.48^2-3.36^2}}$ = 0.08

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

Therefore, the checking is satisfactory.

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

C_1 = 1.77

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

= 1108.8kNm

$\lambda_{LT} = \sqrt{\frac{W_y f_y}{M_{cr}}} = \sqrt{\frac{442.1\times10^3\times355}{1108.8\times10^6}} = 0.38$

$\lambda_{LT,0} = 0.4$

$\beta = 0.75$

$h/b = 597/220 = 2.71$

curve c

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

$\varnothing_{LT} = 0.5[1+\alpha_{LT}(\lambda_{LT}- \lambda_{LT,0})\beta\lambda_{LT}^2]$ = 0.5[1+0.49(0.38-0.4)+(0.75x0.38^2)] = 0.55

$\chi_{LT} = \frac{1}{\varnothing LT+\sqrt{\varnothing_{lT}^2-\beta\lambda_{LT}^2}} = \frac{1}{0.55+\sqrt{0.55^2-0.75\times0.38^2}} = 1$

$M_{b,Rd} = \frac{\chi_{LT} W_{pl,y} f_y}{\gamma_{M1}} = \frac{1.0\times442.1\times10^3\times355}{1.0}$ x10^{-6} = 156.95kNm > M_{Ed} = 17kNm

Therefore, the checking is satisfactory.

Interaction of Axial Force and Bending Moment:

$\lambda_y = 3.36 > 0.4$

$\psi=0$

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

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

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

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

$\frac{140}{389} + 0.77\frac{17}{165.95} \le 1.0$ = 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-(2x220x17.5)+[9.2+2(24)]17.5 = 7011.5mm^2

$\eta h_w t_w$ = 1.0x562x9.8 = 5507.6mm^2 < A_v = 7011.5mm^2

$V_{pl,Rd} = [A_v(f_y/\sqrt{3})/\gamma_{M0}]$ = [7011.5(355/$\sqrt{3}$)/1.0]x10^{-3} = 1437.1kN > V_{Ed} = 16kN

Therefore, the shear resistance checking is satisfactory.

Bending & Shear Interaction:

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

Therefore, the Bending & Shear checking is satisfactory.

Compression Resistance:

$N_{c,Rd} = (Af_y)/\gamma_{M0}$ = (13700x355)x10^{-3} = 4863.5kN

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

Therefore, the Compression Resistance checking is satisfactory.

Bending & Axial Force Interaction:

$0.25N_{pl,Rd}$ = 0.25x4863.5 = 1215.88kN

$(0.5h_w t_w f_y)/\gamma_{M0}$ = [(0.5x562x9.8x355)/1.0]x10^{-3} = 977.6kN

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

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

Bending Moment Resistance:

$M_{pl,Rd}$ = $(w_{py}f_y)/\gamma_{M0}$ = [(442.1x10^{-3}x355)/1.0] x10^{-6} = 156.95kNm > M_{Ed} = 17kNm

Therefore, the Bending Moment Resistance checking is satisfactory.

Out-of-Plane Buckling

h/b = 597/220 = 2.71

t_f = 17.5mm

curve b

α = 0.34

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

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

$\varnothing_z = 0.5[1+\alpha_z(\lambda_z-0.2)\ \lambda_z^2]$ = 0.5[1+0.34(0.46-0.2)+0.46^2] = 0.65

$\chi_z = \frac{1}{\varnothing z+\sqrt{\varnothing_z^2-\lambda_z^2}} = \frac{1}{0.65+\sqrt{0.65^2-0.46^2}}$ = 0.91

$N_{b,z,Rd} = \chi_z A f_y/\gamma_{M1}$ = [(0.91x13700x355)/1.0] x10^{-3} = 4426kN > N_{Ed} = 53kN

Therefore, the checking is satisfactory.

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

C_1 = 1.0

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

= 5925kNm

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

$\lambda_{LT,0}$ = 0.4

β = 0.75

h/b = 597/220 = 2.71

curve c

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

$\O_{LT} = 0.5[1+\alpha_{LT}(\lambda_{LT}- \lambda_{LT,0})\beta\lambda_{LT}^2]$ = 0.5[1+0.49(0.16-0.49)+(0.75x0.16^2)] = 0.43

$\chi_{LT} = \frac{1}{\O LT+\sqrt{\O_{lT}^2-\beta\lambda_{LT}^2}} = \frac{1}{0.43+\sqrt{0.43^2-0.75\times0.16^2}}$ = 1.19

$M_{b,Rd} = \frac{\chi_{LTW_{pl,y}f_y}}{\gamma_{M1}} = \frac{1.19\times442.1\times10^3\times355}{1.0}$ x10^{-6} = 186.72kNm > M_{Ed} = 2kNm

Therefore, the checking is satisfactory.

Interaction of Axial Force and Bending Moment:

C_{mLT} = 1.0

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

$K_{zy} = \max\left[\left(1-\frac{0.1\times0.46}{1-0.25}\frac{53}{4426}\right);\left(1-\frac{0.1}{1-0.25}\frac{53}{4426}\right)\right]$ = max[1;1] = 1

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

$\frac{53}{4426}+1\frac{2}{186.72}\leq 1.0$ = 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

α = 0.21

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

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

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

$\chi_y = \frac{1}{\text{Ø}y+\sqrt{\text{Ø}_y^2-\lambda_y^2}} = \frac{1}{2.4+\sqrt{2.4^2-1.86^2}}$ = 0.26

$N_{b,y,Rd} = \chi_y A f_y/\gamma_{M1}$ = [(0.26x13700x355)/1.0] x10^{-3} = 124.67kN > N_{Ed} = 53kN

Therefore, the checking is satisfactory.

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

$M_{b,Rd}$ = 186.72kNm

Interaction of Axial Force and Bending Moment:

α_{h} = 1

C_{my} = 0.95+0.05α_h = 0.95+0.05(1) = 1

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

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

$$C = \cos\theta = \frac{x_j - x_i}{L} \qquad S = \sin\theta = \frac{y_j - y_i}{L}$$

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

I = 1.188E+06mm^4

A = 1.370E+04mm^2

E = 2.00E+05N/mm^2

Element 1

L = 11000mm

$X_1 = 0$

$X_2 = 0$

$Y_1 = 0$

$Y_2 = 11$

$C = 0$

$S = 1$

$$k = \begin{bmatrix} 2.14 & 0 & -11781.82 & -2.14 & 0 & 11781.82 \\ 0 & 249091 & 0 & 0 & -249090.9 & 0 \\ -11782 & 0 & 8.64\times10^{7} & 11781.82 & 0 & 4.32\times10^{7} \\ -2.14 & 0 & 11781.82 & 2.14 & 0 & 1.18\times10^{4} \\ 0 & -2\times10^{5} & 0 & 0 & 249090.9 & 0 \\ -11782 & 0 & 4.32\times10^{7} & 11781.82 & 0 & 8.64\times10^{7} \end{bmatrix}$$

Element 2

L = 6080mm

X_1 = 0

X_2 = 6

Y_1 = 0

Y_2 = 1

C = 0.9868

S =0.1645

k =

$$\begin{bmatrix} 438876.8 & 73144.02 & -6342.89 & -438876.8 & -73144.02 & -6342.89 \\ 73144.02 & 12203.37 & 38057.32 & -2223.58 & -12203.37 & 38057.32 \\ -6342.89 & 38057.32 & 1.563\times10^{8} & 6342.89 & -38057.32 & 7.816\times10^{7} \\ -438876.8 & -2223.58 & 6342.89 & 438876.8 & 73144.02 & 6342.89 \\ -7.3\times10^{4} & 12203.37 & -38057.32 & 73144.02 & 12203.37 & -38057.32 \\ -6342.89 & 38057.32 & 7.816\times10^{7} & 6342.89 & -38057.32 & 1.563\times10^{8} \end{bmatrix}$$

Element 3

L = 6080mm

X_1 = 0

X_2 = 6

Y_1 = 0

Y_2 = -1

C = 0.9868

S = -0.1645

k =

$$\begin{bmatrix} 438876.8 & -73144.02 & 6342.89 & -438876.8 & 73144.02 & 6342.89 \\ -73144.02 & 12203.37 & 38057.32 & 2223.58 & -12203.37 & 38057.32 \\ 6342.89 & 38057.32 & 1.563\times10^{8} & -6342.89 & -38057.32 & 7.816\times10^{7} \\ -438876.8 & 2223.58 & -6342.89 & 438876.8 & -73144.02 & -6342.89 \\ 7.3\times10^{4} & -12203.37 & -38057.32 & -73144.02 & 12203.37 & -38057.32 \\ 6342.89 & 38057.32 & 7.816\times10^{7} & -6342.89 & -38057.32 & 1.563\times10^{8} \end{bmatrix}$$

Element 4

L = 11000mm

X_1 = 0

X_2 = 0

Y_1 = 0

Y_2 = -11

C = 0

S =-1

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

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]

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

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

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

APPENDIX A4

ELEMENT FORCES OF PORTAL FRAME

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

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

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

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

Element 1

$c = 0$

$s = 1$

$C_1 = 249090.9$

$C_2 = 0.1785$

$L = 11000\text{mm}$

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

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

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

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

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]

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

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

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]

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

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

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]

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

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

APPENDIX B1

LOG FILE

```
/COM,ANSYS RELEASE 12.0.1  UP20090415 19:11:21    04/07/2019
/input,menust,tmp,'',,,,,,,,,,,,,,,,,1
/GRA,POWER
/GST,ON
/PLO,INFO,3
/GRO,CURL,ON
/CPLANE,1
/REPLOT,RESIZE
WPSTYLE,,,,,,,0
/input,menust,tmp,'',,,,,,,,,,,,,,,,,1
/GRA,POWER
/GST,ON
/PLO,INFO,3
/GRO,CURL,ON
/CPLANE,1
/REPLOT,RESIZE
WPSTYLE,,,,,,,0
/TITLE,STEEL FRAME
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,0.924021086472,1
/REP,FAST
/DIST,1,0.924021086472,1
/REP,FAST
~CFACTIV,CIVILFEM,Y
FINISH
~UNITS,SI
/PREP7
!*
ET,1,Beam188
~CFMP,1,LIB,STEEL,EC3,Fe E355
~SSECLIB,1,1,19,59   !UB 914 x 419 x 388
~SSECLIB,2,1,20,31   !UC 356 x 406 x 634
~MEMBPRO,1,EC3,ALL,6.000,1.000,1.000,1.000,0.000,1.000,1.000,1.000,1.000,,1,1.000,1.000,1
~MEMBPRO,1,NLMOD,ALL,0
~MEMBPRO,1,NAME,BEAM
~MEMBPRO,2,EC3,ALL,3.150,1.000,1.000,1.000,0.000,1.000,1.000,1.000,1.000,,1,1.000,1.000,1
~MEMBPRO,2,NLMOD,ALL,0
~MEMBPRO,2,NAME,COLUMN
~MEMBPRO,3,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,3,NLMOD,ALL,0
~MEMBPRO,3,NAME,BEAM 1
~BMSHPRO,1,BEAM,1,1,,,188,1,0,,Beam
~BMSHPRO,2,BEAM,2,2,,,188,2,0,,COLUMN
~BMSHPRO,3,BEAM,1,1,,,188,3,0,,Beam 1
TEMP=150

ELASTIC=2.5E+011

POISSON=0.4

!*

MPTEMP,,,,,,,,

MPTEMP,1,TEMP

MPDE,EX,1

MPDE,NUXY,1

MPDATA,EX,1,,ELASTIC

MPDATA,NUXY,1,,POISSON

!*

DENS=7850

MPTEMP,,,,,,,,

MPTEMP,1,70

MPDE,DENS,1

MPDATA,DENS,1,,DENS
K,1,0,0,0,
K,2,6,0,0,
K,3,12,0,0,
K,4,18,0,0,
K,5,0,3.15,0,
K,6,6,3.15,0,
K,7,12,3.15,0,
K,8,18,3.15,0,
K,9,0,6.3,0,
K,10,6,6.3,0,
K,11,12,6.3,0,
K,12,18,6.3,0,
K,13,0,9.45,0,
K,14,6,9.45,0,
K,15,12,9.45,0,
K,16,18,9.45,0,
K,17,0,12.6,0,
K,18,6,12.6,0,
K,19,12,12.6,0,
K,20,18,12.6,0,
K,21,0,15.75,0,
K,22,6,15.75,0,
K,23,12,15.75,0,
K,24,18,15.75,0,
K,25,0,0,10,
K,26,6,0,10,
K,27,12,0,10,
K,28,18,0,10,
K,29,0,3.15,10,
K,30,6,3.15,10,
K,31,12,3.15,10,
K,32,18,3.15,10,
K,33,0,6.3,10,
K,34,6,6.3,10,
K,35,12,6.3,10,
K,36,18,6.3,10,
K,37,0,9.45,10,
K,37,0,9.45,10,
K,37,0,9.45,10,
K,37,0,9.45,10,
K,37,0,9.45,10,
K,37,0,9.45,10,
K,38,6,9.45,10,
K,39,12,9.45,10,
K,40,18,9.45,10,
K,41,0,12.6,10,
K,42,6,12.6,10,
K,43,12,12.6,10,
K,44,18,12.6,10,
K,45,0,15.75,10,
K,46,6,15.75,10,
```

```
K,47,12,15.75,10,
K,48,18,15.75,10,
K,48,18,15.75,10,
FINISH
/SOL
FINISH
/PREP7
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
SMRT,6
SMRT,5
SMRT,4
SMRT,3
SMRT,2
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,0.924021086472,1
/REP,FAST
/USER,  1
/FOC,   1,
9.22746813428    ,
8.05191966000    ,
5.00000000000
/REPLO
/FOC,   1,
9.22746813428    ,
8.05191966000    ,
5.00000000000
/LIG,   1,1, 1.000,
0.00000000000    ,
0.00000000000    ,
0.00000000000    ,
0.00000000000
/REPLO
/ZOOM,1,RECT,1.05629,-
0.254938 ,0.857472847313 ,-
0.280864184776
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
```

```
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/DIST,1,1.08222638492,1
/REP,FAST
/FOC,   1,
11.0851872184    ,
5.70961747664    ,
5.00000000000
/REPLO
/VIEW,  1, -
0.209411998091E-01,
0.261738623591E-01,
0.999438039640
/ANG,   1, -
0.204248811153E-01
/REPLO
/VIEW,  1, -
0.397588288766    ,
0.297710623071    ,
0.867923923824
/ANG,   1,  -2.93713077349
/REPLO
/VIEW,  1, -
0.663571196959    ,
0.395964692310    ,
0.634732407405
/ANG,   1,  -7.79613724970
/REPLO
/DIST,  1,   19.7474080696
/ANG,   1,   3.60386275030
/REPLO
/DIST,  1,   21.0028504223
/ANG,   1, -0.596137249700
/REPLO
LSTR,      21,      22
LSTR,      22,      23
LSTR,      23,      24
LSTR,      21,      17
LSTR,      17,      13
LSTR,      13,       9
LSTR,       9,       5
LSTR,       5,       1
LSTR,      17,      18
LSTR,      18,      19
LSTR,      19,      20
LSTR,      24,      20
LSTR,      20,      16
LSTR,      16,      12
LSTR,      12,       8
LSTR,       8,       4
LSTR,      13,      14
LSTR,      14,      15
LSTR,      15,      16
LSTR,       9,      10
LSTR,      10,      11
LSTR,      11,      12
LSTR,       5,       6
LSTR,       6,       7
LSTR,       7,       8
LSTR,       1,       2
LSTR,       2,       3
LSTR,       3,       4
LSTR,      24,      48
LSTR,      23,      47
LSTR,      22,      46
LSTR,      21,      45
LSTR,      48,      44
LSTR,      44,      40
LSTR,      40,      36
LSTR,      36,      32
LSTR,      32,      28
LSTR,      45,      41
LSTR,      41,      37
LSTR,      37,      33
LSTR,      33,      29
LSTR,      29,      25
LSTR,      25,      26
LSTR,      26,      27
LSTR,      27,      28
LSTR,       1,      25
LSTR,       2,      26
LSTR,       3,      27
LSTR,       4,      28
LSTR,      47,      43
LSTR,      43,      39
LSTR,      39,      35
LSTR,      35,      31
LSTR,      31,      27
LSTR,      46,      12
LDELE,      55
LSTR,      46,      42
LSTR,      42,      38
LSTR,      38,      34
LSTR,      34,      30
LSTR,      30,      26
LSTR,      48,      47
LSTR,      47,      46
LSTR,      46,      45
LSTR,      22,      18
LSTR,      18,      14
LSTR,      14,      10
LSTR,      10,       6
LSTR,       6,       2
LSTR,      23,      19
LSTR,      19,      15
LSTR,      15,      11
LSTR,      11,       7
LSTR,       7,       3
LSTR,      44,      43
LSTR,      43,      42
LSTR,      42,      41
LSTR,      37,      38
LSTR,      38,      39
LSTR,      39,      40
LSTR,      36,      35
LSTR,      35,      34
LSTR,      34,      33
LSTR,      29,      30
LSTR,      30,      31
LSTR,      31,      32
LSTR,      20,      44
LSTR,      16,      40
LSTR,      12,      36
LSTR,       8,      32
LSTR,      17,      41
LSTR,      13,      37
LSTR,       9,      33
LSTR,       5,      29
/VIEW,  1, -
0.603727540953    ,
0.199243927366    ,
0.771890480380
/ANG,   1,   2.22308795583
/REPLO
FLST,2,36,4,ORDE,12
FITEM,2,1
FITEM,2,-3
FITEM,2,9
FITEM,2,-11
FITEM,2,17
FITEM,2,-28
FITEM,2,43
FITEM,2,-45
FITEM,2,60
FITEM,2,-62
FITEM,2,73
FITEM,2,-84
LMESH,P51X
/UI,MESH,OFF
TYPE,   1
```

```
MAT,      1
REAL,
ESYS,      0
SECNUM,   1
!*
/UI,MESH,OFF
LSTR,      21,      45
LSTR,      17,      41
LSTR,      13,      37
LSTR,       9,      33
LSTR,       5,      29
LSTR,       1,      25
/UI,MESH,OFF
FLST,2,56,4,ORDE,12
FITEM,2,4
FITEM,2,-8
FITEM,2,12
FITEM,2,-16
FITEM,2,29
FITEM,2,-42
FITEM,2,46
FITEM,2,-59
FITEM,2,63
FITEM,2,-72
FITEM,2,85
FITEM,2,-92
LMESH,P51X
LSTR,      18,      42
LSTR,      19,      43
LSTR,      14,      38
LSTR,      15,      39
LSTR,      10,      34
LSTR,      11,      35
LSTR,       6,      30
LSTR,       7,      31
FLST,2,8,4,ORDE,2
FITEM,2,93
FITEM,2,-100
LMESH,P51X
FINISH
/SOL
!*
ANTYPE,0
FLST,2,8,1,ORDE,8
FITEM,2,96
FITEM,2,-97
FITEM,2,103
FITEM,2,109
FITEM,2,115
FITEM,2,-116
FITEM,2,122
FITEM,2,128
!*
/GO
D,P51X, ,0, , , ,ALL, , , , ,
FLST,2,8,1,ORDE,8
FITEM,2,1
FITEM,2,-2
FITEM,2,8
FITEM,2,14
FITEM,2,134
FITEM,2,-135
FITEM,2,141
FITEM,2,147
!*
/GO
FY1=-20000
F,P51X,FY,FY1
FLST,2,96,2,ORDE,6
FITEM,2,109
FITEM,2,-144
FITEM,2,287
FITEM,2,-316
FITEM,2,357
FITEM,2,-386
UDL=4000
SFBEAM,P51X,1,PRES,UD
L, , , , , ,0
FINISH
/POST1
FINISH
/SOL
/STATUS,SOLU
SOLVE
FINISH
/POST1
PLDISP,2
!*
/EFACET,1
PLNSOL, U,SUM, 0,1.0
~CFSET,0,1,LAST,
~PLLSFOR,F,X,-1,
!*
~PLLSSTR,SX,MAX
~PLLSFOR,F,Y,-1,
~PLLSFOR,M,Y,-1,
~PLLSFOR,M,Z,-1,
~CHKSTL,TENSION,-
Z,PARTIAL
~CHKSTL,TENSION,-
Z,PARTIAL
~PLLSSTL,ELM_OK,1,
~CHKSTL,COMPRESS,-
Z,PARTIAL
~CHKSTL,BENDING,-
Z,PARTIAL
~CHKSTL,SHEAR,-
Z,PARTIAL
~CHKSTL,BUCK_CMP,-
Z,PARTIAL
~CHKSTL,BUCK_BND,-
Z,PARTIAL
~CHKSTL,BEND_AXL,-
Z,PARTIAL
~CHKSTL,BEND_SHR,-
Z,PARTIAL
~CHKSTL,BD_AX_SH,-
Z,PARTIAL
~CHKSTL,BUCK_BCM,-
Z,PARTIAL
~PLLSSTL,ELM_OK,1,
~PLLSSTL,ELM_OK,1,
~PLLSSTL,ELM_OK,1,
~PLLSSTL,ELM_OK,1,
~PLLSSTL,ELM_OK,1,
~PLLSSTL,ELM_OK,1,
~PLLSSTL,ELM_OK,1,
~PLLSSTL,ELM_OK,1,
~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
```