SEISMIC LOAD ANALYSIS FOR AN AIRPORT CANOPY STRUCTURE

by

Juma Moses Boyong Apollo

Dissertation submitted in partial fulfilment of
the requirements for the
Bachelor of Engineering (Hons)
(Civil Engineering)

SEPTEMBER 2012

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

SEISMIC LOAD ANALYSIS FOR AN AIRPORT CANOPY STRUCTURE

by

Juma Moses Boyong Apollo

A project dissertation submitted to the

Civil Engineering Program

Universiti Teknologi PETRONAS

in partial fulfillment of the requirement for the

BACHELOR OF ENGINEERING (Hons)

(CIVIL ENGINEERING)

Approved by,

____

(AP. Dr. NARAYANAN SAMBU POTTY)

Date: 11th December, 2012

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

September 2012

CERTIFICATION OF ORIGINALITY

This is to certify that I, **Juma Moses Boyong Apollo** (I/C No: R00019026), am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

____

JUMA MOSES BOYONG APOLLO

Student ID : 12931

I/C No : R00019026

Date : 11th December, 2012

ABSTRACT

Although the Malaysian peninsula is not located along the active seismic zone compared to Sumatra, the tremors due to Sumatran earthquakes have been felt several times. The earthquakes had caused tremor in several cities in the Malaysian Peninsula along the coastal line with Sumatra such as Penang, Port Klang, Selangor and old Klang. Especially the earthquake near Sumatra which happened on 2nd November 2002 had caused panic among residents living in high rise building in Penang prompting thousands of people to run out from their buildings. The paper summarizes the analysis of seismic loadings on the building structures. The seismic analysis is performed in order to assess the force and deformation demands and capacities of the structure. In the analysis, the effect of the provision of gravity, wind loadings on the structure is considered. In this research work, methods such equivalent static analysis and nonlinear dynamic analysis procedure will be used to evaluate the seismic response of the building structure. The structure is modelled and analysed in STAAD Pro software. The test results of each method used in the analysis of structure performance will be discussed and conclusions drawn by comparing the results of the deformation geometry of the graphs and stresses in the members with the prescribed limiting values.

ACKNOWLEDGEMENT

First of all, I take this opportunity to express my sincere and heartfelt gratitude to my supervisor, Assoc. Prof Dr Narayanan Sambu Potty for his expert guidance, monitoring and invaluable suggestions throughout the entire course of this thesis. The blessing, his kind attention and words of motivation given to me by him time to time shall carry me a long way in the journey of life on which I am about to embark. I would also like to give my sincere thanks to the final year project coordinators Dr. Teo Wee and Ir. Idris Othman, lecturers, colleagues, internal as well as external examiners for their valuable time spent in making this project a success. I would like to express a deep sense of gratitude to the entire department of civil engineering, Universiti Teknologi PETRONAS for giving me the opportunity to conduct this research work for my final year project by providing excellent learning environment, facilities and equipment for its success. I am grateful to all my friends and siblings for their constant words of encouragement; it has been an exciting and memorable years of study in this university. Last but not least, words are inadequate in offering my thanks to the most important persons in my life, my parents for their unending blessings, love and for believing in me. They have taught me a lot and I owe all of my achievement to them.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER ONE

INTRODUCTION

1.1 Background of Study

Earthquakes are natural hazards that cause severe damage to or collapse of buildings and other structures. Experience has shown that establishing earthquake resistant regulations in the design of new buildings is a critical safety precaution against earthquake induced damages. In regards to existing structures, seismic evaluation and strengthening is necessary before any sudden earthquake occurrence. The earthquake damage depends on many parameters which include intensity, duration and frequency content of ground motion, geologic and soil condition and sometimes on quality of the construction, etc. when designing a building structure, the design must be such as to ensure that the building has adequate strength, high ductility and must remain as a unit even when subjected to very large deformation.

The study of earthquakes has been an area of great interest in the field of civil engineering. In frequent earthquake prone countries such as Japan, Indonesia and the US, civil engineers have picked interest in this study. In the case of Malaysia, people are living with fear due to recurrence earthquake activities in its neighbourhood Indonesia which is located in the active seismic zone. Malaysia is not located geographically in a designated zone which experience high seismic activity. According to report in the website of "Earthquake Today", in 2004 a pair of massive earthquakes of magnitude 9.1 and 9.3 occurred off the coast of Indonesia causing a fair bit of panic but with minimal damage. However little can be done to diminish direct earthquake effects but we can do much to reduce risks and thereby reduce disasters provided we design and built or strengthen the building structures so as to minimize the losses based on the knowledge of earthquake performance of buildings during earthquakes.

Observation of structural performance of building under simulated earthquake gives us an understanding of earthquakes and their effects on building structures as it can clearly identify the strong and weak aspects of the design, as well as the desirable qualities of materials, techniques of construction, and site selection. The study of building response to earthquake ground motion is an important aspect in the evolution of strengthening measures for building structures while considering the elastic and post elastic behaviour of structures. Structures are generally expected to deform inelastically when subjected to severe seismic loadings, so in this case seismic performance evaluation of the structures is very important and should be conducted considering post elastic behaviour to the structure.

1.2 Problem Statement

1.2.1 Problem Identification

Earthquake as a natural hazard is a sudden and violent shaking of ground which creates lateral forces that causes collapse of structures. These forces are generated by the inertia of building structures as they dynamically response to motion. However, once such seismic loadings are not considered during design and construction of building structures, it may result into collapse or cause severe damage of the structures in case of any earthquake occurrence. Therefore, there is a need to examine building structure response and performance under a simulated seismic ground motion in STAAD Pro software in order to design a building structure that can resist the effect of earthquake motion using the Codes.

1.2.2 Significance of this Project

Through this project, propose variables such as lateral base shear, displacement (demand) and structure's ability to resist seismic loads (Capacity) due to seismic ground motion can be investigated and analysed through simulation using STAAD Pro software. However, more variables can be further study in this project.

1.3 Objective

The main objectives of the study are:

1. To analyse the safety of a long span existing building structure under seismic loading using the different analysis procedures.
2. To analyse the response of long span steel frame building in Malaysia under seismic loading using Equivalent static analysis
3. To Study the behaviour of long span building structure under seismic load using nonlinear dynamic analysis.

1.4 Scope of study

The gravity loads, wind loads and earthquake loads on long span structure will be calculated using the codes e.g. ASCE, Eurocode, and Malaysian Code (MS 1553) only. The modelling and analysis of the long span building structure will be done using STAAD Pro software. The structure build is a steel structure and a simplified model of the airport canopy will be used for the analysis. The loadings evaluated will correspond to that in Malaysia.

1.5 Relevancy of this project

Since earthquake loading causes cracks or severe damage to building structures, this study will give us a break through in estimating the maximum seismic design load for building structures. By knowing the seismic peak ground acceleration that has most occurrences, new building structures that are resistant to earthquake induced damage and critical for the safety of a building structure under seismic loads can be designed.

1.6 Feasibility of this project

This project is encompassing research and simulation work. The Simulation related work is based on the use of STAAD Pro to model and analyse the structure. Similarly, software such as ETAP can be used in modelling of the structure. However, further research work is needed for more information on the use of codes for the design of the building structures e.g. Eurocode, ASCE Code, British Code, and Malaysian Code (MS). This project can be achieved within 8 months given that resources for the project are available. The objective can be achieved if the procedures are closely followed.

CHAPTER TWO

LITERATURE REVIEW

2.1 Introduction

The load analysis of a structure such as canopy is very complex. Long span structures have to support gravity loads but however when lateral forces are applied such a wind and seismic loading, additional structural elements are required to sustain them. These additional forces applied can have an effect on the integrity of the whole structure. Most structure performance such as canopy under lateral loadings such as seismic loads is much dependent on the intensity of the ground shaking caused by the earthquake. Besides, factors such as strength of shaking, type of the soil and the frequency of resonance play a part.

This review also focuses on the provisions of various design codes (ASCE, Eurocode and MS 1553) regarding gravity, wind and seismic loads for the canopy structure design, compare their performance variation in structure design and also elaborate briefly on previous work performed by other researchers in regards to seismic loads analysis using the linear and nonlinear analysis methods in determining the performance of a structure under seismic ground motion. The literature review is organised into the following sections:

2.2 Canopy structure

A canopy structure is simply a long span structure which is term as a "special structure". Bradshaw (2002) refers to innovative long span structural system, primary roofs and enclosures to house human activities. The special structures include space frame or grids; cable and strut and tensegrity; air-support or air-inflated; self-erecting and deployable; cable net; tense membrane; geodesic domes; folded plates; and thin plates. Thin shells and tension membranes are considered form-resisting structures, as they resist loads by virtue of their shape. Neither will function if flat, and both carry loads predominantly through in-plane stresses rather than by bending, granted that thin shells bend as well as compress. Other structures resist loads mostly in flexure.

Canopies composed of lightweight frame structure over which cover is attached. Most of the canopies consist of fabric stretched over and secured to a fixed metal frame that is secured by laces or screws. These frames may be welded, bolted or otherwise connected. Other canopies that consist of individual fabric panels can be attached using staple-in method. Still others consist of rollers and lateral arms that can be retracted manually or automatically. It should be noted, however, that the possible combinations of styles, configuration and colours are limitless.

2.3 Forms of canopy structures

The classification of canopies structures is mostly derived from their style and materials they are made from. These canopy structures include the following.

2.3.1 Tensile fabric canopies

A tensile canopy is held together by a strong fabric or a network of wire or cable. The tension that these elements are under supports the entire structure. Normally lightweight frames are used for the construction of these structures. Such a tensile structure can take a dome, modelled on geodesics or be cone or saddle shaped. Buildings as large as stadiums and airports terminal have been built using tension-based structural systems. Tensile fabric canopy as shown in (fig 2.1) uses powder coated steel frame supporting a heavy duty fabric roof membrane.

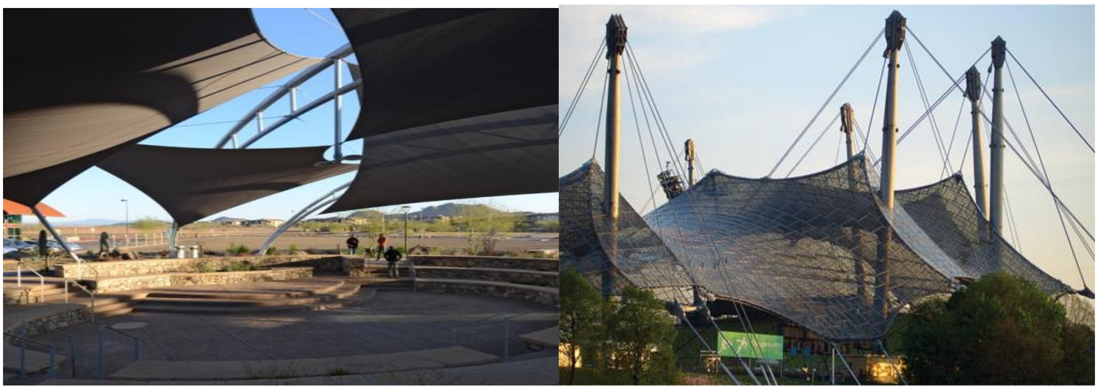

Figure 2.1 Tensile fabric canopy structure (Munich Olympic Stadium, 2012)

2.3.2 Steel canopies

Steel canopies styles include tapered trellis canopies perfect for covering large space on a budget. Steel canopies as shown in (fig 2.2) are made from a galvanised steel frames which need no maintenance. Alternatively they can be polyester powder coated in colour.

Figure 2.2 Steel canopy structure (sources: fordingbridge, 2012)

2.3.3 Timber canopies

Timber canopies as shown in (fig 2.3) are canopies made from laminated timber frame or a lightweight alternative. The frame style include curved barrelled vault, mono-pitched canopies. The roof coverings which is a fire retardant fabric with high light transmission or an opaque 16mm triple wall polycarbonate.

Figure 2.3 Timber canopy structure (sources: Innovarchi & fordingbridge, 2012)

2.3.4 Barrel vault canopies

Barrel vault canopies as shown in (fig 2.4) are cylindrical (arched) form roof canopies. These free standing canopies are ideal for covering larger area. It can be designed in single or multi-span barrel vault made using laminated timber or steel.

Figure 2.4 Barrel vault canopy structure (Source: ABLE CANOPIES, 2012)

2.3.5 Mono-pitched canopies

Mono-pitched canopies as shown in (fig 2.5) are canopies with single sloped roof. They are used to cover spaces alongside existing buildings. Mono-pitched canopies can be used also to create additional shade preventing buildings getting too hot. These types of canopies are made of laminated timber, steel and glass.

Figure 2.5 Mono-pitched canopy structure (Sources: Broxap, 2012)

2.3.6 Cantilever canopies

A cantilever canopy is a free standing structure which is mostly constructed from a timber frame with a curved roof. This type of structure can be fitted with a water proof tensile fabric or polycarbonate roofing, giving a flexibility to choose right materials. This type of canopies is ideal for site where limited number of posts is required or where it is difficult to place foundation near existing building. Cantilever canopies as shown in (fig 2.6) are design in structural glue-laminated timber, steel and tensile fabric.

Figure 2.6 Cantilever canopy structure (Sources: Abacus shade, 2012)

2.3.7 Dual-pitched Canopies

Dual-pitched canopies as shown in (fig 2.7) are canopies with double slope roof on both sides of a central ridge e.g. gambrel roof. It provides better weather protection. Most of dual pitched canopies are made of steel or aluminium framed, glazed with multiwall polycarbonate non-fragile roof panels.

Figure 2.7 Dual pitch free standing canopy (sources: Clovis & Broxap, 2012)

2.4 Example of canopy structures in Malaysia

The KLIA long span canopy roof and UTP long span canopy roof as shown in (fig 2.8.a) and (fig 2.8.b) are the typical canopy structures in Malaysia. The Kuala Lumpur International Airport roof structure is formed of inclined bow-string trusses that span the width of the building at 9.8 m intervals. The trusses are supported at points along their length by exposed pin-jointed raking tubular columns that extend up, in splays of four, from the tops of cantilever concrete columns. However the UTP canopy, 2004(UTP, 2012), steel design, covers the new academic complex.

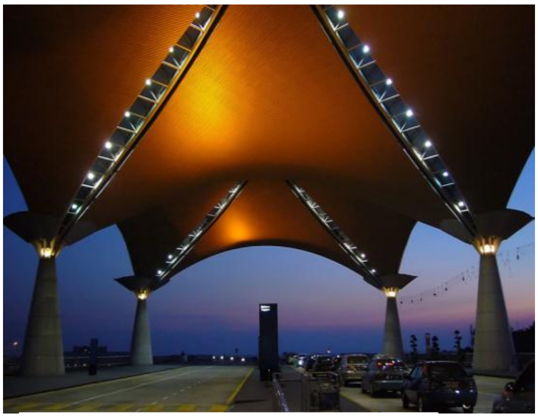

Figure 2.8.a KLIA Canopy

Figure 2.8.b UTP Canopy

2.5 Materials for making canopy structures

The typical material used in the construction of a permanent or temporary canopy structures include the following

- Membrane (fabric) e.g. PTFE (polytrafluoroethylene expanded fibre), coated woven fibreglass, and PVC composite membranes
- Steel elements e.g. steel rods and cables used by long span tension cable structures to support horizontal roof surfaces
- Timber elements e.g. laminated timber frames
- Aluminium elements e.g. aluminium frames and cables

2.6 Types of loadings on canopy structure

There are practically different types of loads and forces acting on the structure. These loads include gravity loads i.e. Dead loads and live loads, wind loads, snow loads, earthquake forces, shrinkage, creep and temperature effects etc. This primary load types are used to analyse a structure for various parameters like span moments, end moments, shear, torsion and deflections. In practice, in order to capture the critical load patterns on a structure, engineers have limited themselves to the use of standards structural engineering codes such as ASCE05/UBC/EUROCODE and MS 1553 in Malaysia.

2.6.1 Gravity loads

(a) ASCE CODE

1. Dead Load:

Dead load simply includes weight of all materials of construction incorporated into the building e.g. walls, floors, roofs, ceiling etc.

2. Live Load:

Live load results from the use and occupancy of the building. On roofs, live loads results during maintenance by workers, equipment, and material. Enclosed Structures when construction is complete are usually design for roof live load of 0.89 kN and 0.98 kN (AASHTO, 1996).

For flat, pitched, and curved roofs shall be designed for a roof live loads expressed by the equation.

$L_r = 0.96R_1R_2$……………………………….…………………….…….2.1

Where: $0.58 \leq L_r \leq 0.96$ (L_r is roof live load (kN/m^2))

The reduction factors R_1 and R_2 are determined from:

$$R_1 = \begin{cases} 1 & for\ A_t \ \leq 18.58\,m^2 \\ 1.2 - 0.01076\,for\ 18.58A_t & for\ \ 18.58\,m^2\ < A_t < 55.74\,m^2 \\ 0.6 & for\ A_t \ \geq 55.74\,m^2 \end{cases}$$

At = tributary area in (m^2) supported by any structural member.

$$R2 = \begin{cases} 1 & for\ \ F \leq 4 \\ 1.2 - 0.05\,F & for\ 4 < F < 12 \\ 0.6 & for\ \ F\ \geq 12 \end{cases}$$

Where for a pitched roof, F (m): F = 0.12 x slope (%) and for an archor dome, F = Rise to span multiplied by 32.

(b) EUROCODE

1. Permanent Action (Self-weight), *G*

Act throughout a given reference period and for which the variation on magnitude with time is negligible, for which the variation is always on the same direction (monotonic) until the actions attains a certain limit value.

2. Variable Load (Imposed loads)

Imposed loads on building are those arising from occupancy. These loads can be on floors, beams and roofs. Imposed design loads are mostly applied at the unfavourable part of influence area of the action effects considered. The values of actions of imposed loads on;

Roof**:** Assumed category, H, (is a roof not accessible except for normal maintenance and repair. Refer to Eurocode (table 6.10 imposed loads on category H)

Note: For category H, q_k (uniformly distributed loads) may be selected within the range of (0.0 - 1.0) kN/m^2 and Q_k (Concentrated loads) may be selected within the range 0.9 kN to 1.5 kN (Recommended value for q_k = 0.4 kN/m^2 and Q_k = 1.0 kN)

Note: q_k may be varied by the national Annex dependent upon the roof slope. Refer to section EC-1 (section 6.3.4)

Floors and beams**:** Values assigned are based on building categories. The categories of loaded areas as specified in (table 2.1) shall be designed by using characteristic values q_k and Q_k.

For example: Category, A, areas for domestic and residential activities, B is areas for office buildings.

Table 2.1 Imposed loads on floor

Categories of loaded areas	q_k (kN/m^2)	Q_k (kN)
A	1.5 – 2.0	2.0 – 3.0
B	2.0 – 3.0	1.5 – 4.5

2.6.2 Wind loadings

(a) ASCE CODE

The design wind loads for buildings and other structures, including the main force-resisting system, component and cladding elements shall follow the simplified procedures based on (ASCE).

Step 1: Determining the basic wind speed, V, and wind directionality factor k_d with accordance to section 6.5.4 (table 2.2). Assuming, the wind come from any direction e.g. the estimation of basic wind speeds is based on regional climate data.

Table 2.2 Wind directionality factor

Structural Type	Directionality Factor, k_d
Buildings Main wind force resisting system Components & cladding	 0.85 0.85
Arched roof s	0.85
Chimneys, tanks, and similar structures Square Hexagonal Round	 0.90 0.95 0.95
Solid Signs	0.85
Open Signs & lattice framework	0.85
Trussed towers Triangular, square, rectangular Other sections	 0.85 0.95

Step 2: Determine the importance factor, I, with accordance (ASCE)

Table 2.3 Importance factors, I,

Category	Non Hurricane prone regions & Hurricane prone regions with V = 85-100 mph and Alaska	Hurricane prone regions with V >100 mph
I	0.87	0.77
II	1.00	1.00
III	1.15	1.15
IV	1.15	1.15

Step 3: Determine the exposure category or exposure categories and velocity pressure exposure coefficient k_z, k_h (determine for each wind direction)

This reflects the characteristic of ground roughness and surface irregularities for each site. This is determined from section (6.5.6.3). Velocity pressure q_h which is an exposure resulting from the highest wind loads from any wind direction. The external pressure coefficient GC_{pf} is given in (table 2.4). The velocity pressure exposure base on the coefficient based on the exposure category in section (6.5.6.3) k_z or k_h is determined from ASCE.

Step 4: Determine the topography factor, k_{zt}, if applicable according to (section 6.5.7)

$k_{zt} = (1+ k_1k_2k_3)^2$……………………..…………………………….……….2.2

Where; k_1, k_2 & k_3 are given in figure (6-4).

Step 5: Determine the Gust effect factor G, or G_f, as applicable with accordance to ACSE. (Section 6.5.8.) . For rigid structures (G is taken as 0.85) as in section 6.2 or calculated from the formula

$G = 0.925 \left(\frac{1+1.7\, g_Q I_{\check{z}Q}}{1+1.7\, g_v I_{\check{z}}} \right)$, $I_{\check{z}} = C\ (33/\check{z})^{1/6}$………………...…………..…2.3

$I_{ž}$ - Intensity of turbulence height at (ž)

Ž - Equivalent height of the structure

g_Q &g_v are taken as 3.4

Q - Background response

$$Q = \sqrt{\frac{1}{1+0.63\left(\frac{B+h}{L_{ž}}\right)^{0.63}}} \quad \ldots\ldots 2.4$$

Where: *B* and h are define in (section 6.3)

$L_{ž}$ - Integral length of turbulence at the equivalent height

$L_{ž} = l\,(ž/33)ἑ$

l& ἑ constants listed in table (6-2)

Step 6: Determine the enclosure classification in accordance to (section 6.5.9). In order to determine the internal pressure coefficient, all the buildings shall be classified as enclosed, partially enclosed or open as specified in (section 6.2)

Step 7: Determine the internal pressure coefficient, GC_{pi}, in accordance to ASCE (section 6.5.11.1)

The internal pressure coefficients, GC_{pi}, is determine from (tab 2.4) base on the building enclose classifications determine from (section 6-5-9)

Table 2.4 Values of internal pressure coefficient, GCpi,

Enclose Classification	GC_{pi},
Open buildings	0.00
Partially Enclosed Buildings	∓ 0.55
Enclosed Buildings	∓0.18

Notes: Plus and minus signify pressures acting towards and away from the internal surfaces, respectively. The values of GC_{pi}, are used as q_h or qz as specified above.

Step 8: Determine the external pressure coefficients, C_p or GC_{pf} or force coefficient, C_f, as applicable in (section 6.5.11.2)

The external pressure coefficients for main wind force resisting systems C_p are given in figures (6-6, 6-7, and 6-8) page (51-53) and GC_{pf} values are given in figure (6-10) for low-rise buildings.

Step 9: Determine the velocity pressure q_p or q_h as applicable in ASCE (section 6-5-10). The velocity, q_p, is analyse at height z is calculated from equations

$q_z = 0.613\ k_z k_{zt} k_d V^2\ I$ (N/m2); V in m/s…………………………….……2.5

k_d = wind directional factor

k_z = velocity pressure exposure coefficient

k_{zt} = topography factor

Step 10: Determine the design wind pressure, P, or design wind loads, F, with accordance to ASCE (section 6.5.12 and 6.5.13.) the P value is calculated from equations:

For rigid building of all height:

$P = qGC_p - q_i\ (GC_{pi})\ N/m^2$…………………………………..…………2.6

For flexible buildings

$P = qG_fC_f - q_i\ (GC_{pi})\ N/m^2$………………………………..……………2.7

Where:

$q = q_z$ for windward walls elevated at height z above the ground

$q = q_h$ for leeward walls, side walls and roofs, elevated at height h

$qi = q_h$ for wind ward walls, side walls, lee wards walls, and roofs of enclosed building.

$qi = q_z$ for positive internal pressure evaluation in partially enclosed building at height z.

For low-rise building

$$P = q_h\,[(GC_{pf}) - (GC_{pi})]\ N/m^2 \qquad \text{……2.8}$$

The design wind load on open buildings and other structures is given by;

$$F = q_z\, GC_f A_f\ (N) \qquad \text{……2.9}$$

qz = velocity pressure analysis at height z of the centroid of area, A_f.

G = guest effect factor

C_f = net force coefficient from figure (6-18 through 6-22)

A_f = Projected area normal to the wind (m^2)

(b) EUROCODE

Wind action is classified as variable fixed action (EN 1990.4.1.4). It is a set of pressure or forces whose effects are equivalent to the extreme effects of the turbulent. The effect of the wind on the structure depends on the size, shape and dynamic properties of the structure.

Step 1: Determination of the basic velocity

q_p = velocity pressure, C_sC_d = structural factor, V_m = Mean wind velocity, V_b = basic wind velocity, $V_{b,0}$ = Fundamental value of the basic wind velocity, C_{dir} = directional factor, C_{season} = seasonal factor.

$$V_b = C_{dir}.\, C_{season}.\, V_{b,0} \qquad \text{……2.10}$$

$V_{b,0} = V_b\, .C_{prob}$ (C_{prob} is probability factor)

$$C_{prob} = \left(\frac{1-k\,.In\left(-In\,(1-p)\right)}{1-k.\ In\left(-In\,(0.98)\right)}\right)^n \qquad \text{……2.11}$$

(Recommended values k = 0.2 and n= 0.5)

Step 2: Mean wind Velocity: V_m (z) at height z

$V_m (z) = C_r (z)\, C_o (z)\, V_b$ ……………………….……..………...……2.12

C_r (z) is the roughness factor and C_o (z) is orography factor

Step 3: Roughness Factor:

$C_r (z) = k_r.\ln(\frac{z}{z_o})$ for $z_{min} \leq z \leq z_{max}$

$C_r (Z) = C_r (Z_{max})$ for $z \leq z_{max}$

$K_r = 0.19\, (\frac{z_0}{z_{0,II}})^{0.07}$ ……………………………………….…………2.13

$z_{0,\,II}$ = 0.05 (Terrain category II, table 4.1)

z_{min} = minimum height (Table 4.1) and z_{max} = is taken as 200m

Step 4: Terrain Orography: Effect neglected when the average slope of the upwind terrain is less than 3^0

Step 5: Wind turbulence: I_v (z)

$\delta_v = k_r.\, V_b.\, k_1$

k_r = Terrain factor, k_1 = turbulent factor (k_1= 1.0)

$I_v (z) = \frac{\delta_v}{V_{m\,(z)}} = \frac{k_1}{C_{0\,(z)}\, In\, (\frac{z}{z_0})}$ for $z_{min} \leq z \leq z_{max}$…………..…………2.14

$I_v (z) = I_v (z_{min})$ for $z < z_{min}$; z_0 is the roughness length (Table 4.1)

Step 6: Peak velocity pressure: q_b (z) at height z

$q_p (z) = [1+ 7.I_v (z)].\frac{1}{2}\rho V_m^2(z) = C_e(z).q_b$ …………………..………..…2.15

C_e = exposure factor

$C_e (z) = \frac{q_{p (z)}}{q_b}$ (for flat terrain $C_0 (z) = 1.0$

Step 7: Basic velocity Pressure:

$q_b = \frac{1}{2} \rho_{air}. V_b^2$……………………………………………….…...……….2.16

Where: $\rho_{air} = 1.25 kg/m^3$ (density of air)

Wind Action: The wind action taken into account is both external and internal wind pressure on the structure.

Step 8: Wind pressure on external surface (W_e) of structure.

$We = q_p (z). C_{pe}$…………………………………………..……...……2.17

Where: $q_p (z_e)$ - Peak velocity pressure

z_e - Reference height for external pressure (sec-7)

C_{pe} – Pressure coefficient for external pressure (sec-7)

Step 9: Wind pressure on internal surface (W_i) of structure.

$W_i = q_p (zi). C_{pi}$………………………………………………………….2.18

Where: q_p (zi) - Peak velocity pressure

z_i - Reference height for internal pressure (sec-7)

C_{pi} – Pressure coefficient for internal pressure (sec-7)

Step 10: Wind load (W) = $(Cpe + C_{pi}) q_p S$

(c) MALAYSIAN STANDARD (MS 1553: 2002)

When determining wind actions (W) on structure elements of structure or buildings using MS 1553: 2002, the following procedure are taken into consideration;

Step 1: Determine the site wind speed section 2.2

The site wind speeds V_{sit} is normally define at the level of average roof height above ground given by the expression:

$$V_{sit} = V_{s.} (M_d) (M_{z, cat}) (M_s) (M_h) \ldots\ldots 2.19$$

Where: $V_{s.}$ = 33.5 m/s zone I and 32.5 m/s zone II (MS, fig. 3.1) reference based on 50 years return period and has been recommended for zone I and Zone II.

M_d = 1; $M_{z,}$ cat is terrain /height, M_h is hill shape multiplier and M_s is the shielding multiplier (Section 4).

Step 2: Determine design wind speed from the site wind speeds (section 2.3)

In MS 1553:2002, the building design winds speeds, $V_{des,}$ is taken as the maximum site wind speed, V multiplied by the importance factor, *I*, obtained from table (table 3.2).

$$V_{des} = V_{sit} \text{ x } I \ldots\ldots 2.20$$

Step 3: Determine design wind pressures and distributed forces (section 2.4)

The design wind pressure for structures and parts of structures is determined from the equation:

$$\rho = (0.5_{\rho\ air}) [V_{des}]^2 C_{fig} C_{dyn} \rho_a \ldots\ldots 2.21$$

Where: ρ_{air} = 1.225 kg/m^3

C_{fig} = Aerodynamic shape factor (section 5)

C_{dyn}= Dynamic response factor taken as 1, unless the structure is wind sensitive, the values are defined in (section 6)

E.g. free standing hoardings, walls canopies and roof, the aerodynamic shape factors C_{fig} is determined from (appendix D)

$C_{fig} = C_{p,n}K_a\ K_lK_p$ for pressure normal to surface;

$C_{fig} = C_f$ for frictional drag forces.

Step 4: Calculate wind actions (section 2.5)

The structure are designed to withstand wind forces derived by considering wind actions from no fewer than four critical orthogonal directions aligned to the structure.

Actions effects, the forces, F, in (N) on a building element such as wall and roof, is determined from the pressures applicable to the assumed areas as

$$F = \Sigma\ \rho_z A_z \quad \text{………………………………………………………………}2.22$$

Where; ρ_z is the design wind pressure at a height z. ($\rho_e - \rho_i$) for enclosed buildings or (ρ_n) where net pressure is applicable. Given ρ_e, ρ_i, ρ_n are the external, internal and net pressure respectively.

A_z is the area of a structure or a part of structure (m^2) at height z upon which the pressure at the height (ρ_z) acts.

2.7 Earthquakes

Earthquakes are the sudden release of energy occurring from the collision or shifting of crustal plates on the earth's surface or from the fracture of stressed rock formations in that crust. This released of energy results in the earth shaking, rocking, rolling, jarring and jolting; having the potential to cause minimal to great damage to structures. Ground acceleration is the most important measure of earthquake motion from an engineering point of view, for it is directly related to the earthquake force transmitted to the structure. To measure strong ground accelerations, accelerographs must be in place in the region of strong motion before the earthquake occurs.

2.8 Effects of earthquakes on structures

The earthquake shakes the underlying ground in all direction; a building gets thrown from side to side. That is, while the ground is violently moving from side to side, the building tends to resist the motion. This causes strong vibrations of the structure with resonance phenomena between the structure and the ground, and thus large internal forces. This frequently results in plastic deformation of the structure and substantial damage with local failures and, in extreme cases, collapse. The effects of an earthquake on a building are primarily determined by the time histories of the three ground motion parameters; ground acceleration (a_g), velocity (v_g), and displacement (d_g), with their specific frequency contents. Looking at the example of the linear horizontal ground motion chart of an artificially generated (Valais Quake), it is clear that the dominant frequencies of acceleration are substantially higher than those for velocity and much higher than those for displacement. The soil also has a period varying between 0.4 and 1.5 sec, very soft soil being 2.0 sec. Soft soils generally have a tendency to increase shaking as much as 2 to 6 times as compared to rock. Also, the period of the soil coinciding with the natural period of the building can greatly amplify acceleration of the building and is therefore a design consideration.

$$F_{Inertial} = \text{ma}$$

$$F_{\text{Inertial}} = \text{mass (weight of the building) x acceleration (ground acceleration)}$$

The aforementioned seismic measures are used to calculate forces that earthquake impose on buildings. Ground shaking (pushing back and forth, sideways, up and down) generates internal forces within buildings called the Inertial Force (F_{Inertial}), which in turn causes most seismic damage. The greater the mass, the greater the internal inertial forces generated. Lightweight construction with less mass is typically an advantage in seismic design. Greater mass generates greater lateral forces, thereby increasing the possibility of columns being displaced, out of plumb, and/or buckling under vertical load. Earthquakes generate waves that may be slow and long, or short and abrupt. The length of a full cycle in seconds is the Period of the wave and is the inverse of the Frequency. All objects, including buildings, have a natural or fundamental period at which they vibrate if jolted by a shock. The natural period is a primary consideration for seismic design, although other aspects of the building design may also contribute to a lesser degree to the mitigation measures. If the period of the shock wave and the natural period of the building coincide, then the building will "resonate" and its vibration will increase or "amplify" several times.

2.9 Building code requirement for earthquake resistance

Seismic building codes usually specify laterals design forces in a formula that involves the seismicity of the region; the importance of the occupancy; the type, the period, and weight of the structure, and sometime soil properties of the site. Buildings codes are drafted to meet the demands of the expected shaking in a given region that are summarized by seismologists and earthquake engineers in hazards maps. EN 1998 Eurocode 8 “Design of structures for earthquake resistance” applies to design and construction of buildings and other civil engineering works in seismic regions. Its purpose is to ensure that in the event of earthquakes; human life is protected, damages are limited, and structural importance for civil protection remains operational.

	Parts
EN 1998-1 Part 1	General rules, seismic actions and rules for buildings
EN 1998-2 Part 2	Bridges
EN 1998-3 Part 3	Assessment and retrofitting of buildings
EN 1998-4 Part 4	Silos, tanks and pipelines
EN 1998-5 Part 5	Foundations, retaining structures and geotechnical aspects
EN 1998-6 Part 6	Towers, masts and chimneys

EN 8 clearly covers common structures and, although its provisions are of general validity, special structures, such as nuclear power plants, large dams or offshore structures are beyond its scope. Its seismic design should satisfy additional requirements and be subject to complementary verifications.

(a) ASCE CODE

Step 1: Determine the ground motion spectral response acceleration.

S_s = Ground acceleration at short period of 0.2 sec

S_1 = Ground acceleration at longer period of 1.0 sec (see figures 20.1 through 20.14) respectively.

Step 2: Determine "Site Class": Site class is based on seismic shear wave velocity, V_s, travelling through the top 100ft (30m) of the site profile. Seismic load guide provisions (ASCE 7-05)

Step 3: Determine "Maximum Considered Earthquake" Spectral Response**:**

The maximum considered earthquake spectral response acceleration for short period (S_{MS}) at 1sec (S_{M1}) adjusted for site class effect is determine from equations

$S_{MS} = FaSs$……………………………..……………………………….2.24

$S_{M1} = FvS_1$……………………………………………....……………..2.25

Where: S_s = the mapped MCE spectral response acceleration at short period.

S_1 = the mapped MCE spectral response at a period of 1 sec.

Where site coefficients F_v and F_a are defined in (table 2.5) and (table 2.6).

Table 2.5 Site Coefficient, F_a

Site Class	Mapped Maximum Considered earthquake spectral response acceleration parameter at short period				
	$S_s \leq 0.25$	$S_s \leq 0.5$	$S_s \leq 0.75$	$S_s \leq 1.0$	$S_s \leq 1.25$
A	0.8	0.8	0.8	0.8	0.8
B	1.0	1.0	1.0	1.0	1.0
C	1.2	1.2	1.1	1.0	1.0
D	1.6	1.4	1.2	1.1	1.0
E	2.5	1.7	1.2	0.9	0.9
F	See section 11.4-7				

Table 2.6 Site Coefficients, F_V

Site Class	Mapped Maximum Considered earthquake spectral response acceleration parameter at 1 sec period				
	$S_1 \le 0.25$	$S_1 = 0.5$	$S_1 = 0.75$	$S_1 = 1.0$	$S_1 \ge 1.25$
A	0.8	0.8	0.8	0.8	0.8
B	1.0	1.0	1.0	1.0	1.0
C	1.7	1.6	1.5	1.4	1.3
D	2.4	2.0	1.8	1.6	1.5
E	3.5	3.2	2.8	2.4	2.4
F	See section 11.4-7				

However, the design earthquake spectral acceleration parameters at short period S_{DS} and at 1 sec period S_{D1} are determined from equations.

$S_{DS} = (2/3\ S_{MS})$……………………………………..…………………...2.26

$S_{D1} = (2/3\ S_{M1})$……………………………………………………….…2.27

Step 5: Determine "Response Modification Coefficient" R: Check the Value of R from (ASCE)

Step 6: Determine the Effective Seismic Weight of Structure "W":

W = (Total dead loads of structure) "+ "……..

1. In areas used for storage, a minimum of 25% of the reduced floor live load (floor live load in public garages and open parking structures need not be included)
2. Where an allowance for partition load is included in the floor load design, the actual partition weight or a minimum weight of $0.48kN/m^2$ of floor area, whichever is greater.
3. Total operating weight of permanent equipment

Step 7: Determine Occupancy Importance Factor "*I*".

Table 2.7 Occupancy Importance Factors

Seismic Use Group	*I*
I	1.0
II	1.25
III	1.5

Step 8: Determine Seismic Base Shear "V".

V = CsW……………………………………………………………… 2.28

Where: $C_s = \frac{S_{DS}}{\left(\frac{R}{I}\right)}$ (Use the largest C_s), $Cs = \frac{S_{DS}}{T_{a}\left(\frac{R}{I}\right)}$

$T_a = C_t h_n^{x}$ (T_a = approx. fundamental building period, sec)

Table 2.8 Value of C_t and x

Structure Type	C_t	x
Steel moment-resisting frames	0.028	0.8
Concrete moment-resisting frames	0.016	0.9
Eccentricity braced steel frames	0.030	0.75
All other structural system	0.020	0.75

$C_s = \frac{0.5S_1}{\left(\frac{R}{I}\right)}$ If $S_1 > 0.6$ g, $C_s > 0.01$

Step 9: Determine vertical distribution of Seismic Shears.

$$F_x = C_{vx}V; \quad \text{Where: } C_{vx} = \frac{W_x h_x}{\sum_{i=1}^{n} W_i h_i} \quad ,$$

Where: h is height above base (ft); W is portion of weight at that level; C is vertical distribution factor and V is total design lateral force or shear at the base of the structures (kN)

(b) EUROCODE

This method is applied for buildings whose response is not significantly affected by contribution from modes of vibration higher than the fundamental mode in the principle direction. Given below is procedure for seismic load analysis.

Step 1: Conduct appropriate investigations to verify and identify the ground conditions

Step 2: Identification of ground types e.g. ground type A, B, C, D, and E describe by the soil stratigraphic parameters and parameters given in table below. In the table, the site is classified according to the average shear Wave velocity, Vs, 30 if the value available otherwise you can use N_{STP}

$$Vs, 30 = \left(\frac{30}{\sum_{i=1,N} \frac{h_i}{v_i}} \right) \quad \ldots\ldots 2.29$$

Where; h_i = thickness (m) and V_i = shear-wave velocity (at shear strain level of 10^{-5} or less), i-th formation of layers, in total of N, existing in the top 30 m

Table 2.9 Grounds types

Site class	Soil profile name	Average properties in the upper 30 m		
		Shear wave velocity, V_s (m/s)	N_{SPT} (blows/30cm)	Undrained shear strength. C_u (kPa)
A	Rock or thin (< 5m) soil	$800 < V_s$	-	-
B	Very dense or stiff soil	$360 < V_s \leq 800$	N > 50	$C_u > 250$
C	Dense or stiff soil	$180 < V_s \leq 360$	15-50	$70 < C_u \leq 250$
D	Loose or salt to firm soil	$100 < V_s \leq 180$	$5 < N \leq 15$	$20 \leq Cu \leq 70$

Step 3. Selection of the seismic zone in the national annex. Where there is low seismicity cases in the design ground acceleration on type A ground, ag is not greater than 0.08g (0.78m/s) or those where the product ag,S is not great are than 0.1g (0.98m/s).

Step 4. Determination of base shear force F_b for each horizontal direction.

$$F_b = S_d(T_1).m.\lambda \qquad 2.30$$

Where;

$S_d(T_1)$ is the ordinate of the design spectrum at period T_1

T_1 is the fundamental period of vibration of the building

m is the total mass of the building computed according to 3.2.4(2)

λ is the correction factor, the value of λ = 0.85 if $T_1 \leq 2T_c$ or a building having more than 2 storeys λ = 1.0

$$0 \leq T \leq T_B : S_d(T_1) = a_g . \text{S}. \left[\frac{2}{3} + \frac{T}{T_B} . (\frac{2.5}{q} - \frac{2}{3})\right] \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots.. 2.31$$

$$T_B \leq T \leq T_C : S_d(T) = a_g . \text{S}. \frac{2.5}{q} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 2.32$$

$$T_C \leq T \leq T_D : S_d(T) \begin{cases} = a_g . S . \frac{2.5}{q} . [\frac{T_C}{T}] \\ \geq \beta . a_g \end{cases} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 2.33$$

$$T_D \leq T : S_d(T_1) \begin{cases} = a_g . S . \frac{2.5}{q} \left[\frac{T_C T_D}{T^2}\right] \\ \geq \beta . a_g \end{cases} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 2.34$$

Where;

S_e(T) is the elastic response spectrum

T is the vibration period of SDOF

a_g is design ground acceleration on type A ground ($a_g = \lambda_1 a_g$ R)

T_B is the lower limit of the period of the constant spectral acceleration branch

T_C is the upper limit of the period of constant spectral acceleration branch

T_D is the value defining the beginning of the constant displacement response range of the spectrum

S is the soil factor

η is the damping correction factor with reference value η = 1 for 5% viscous damping

The value of the periods $T_B, T_C and\ T_D$ and of the soil factor S describing the shape of the elastic response depends upon the ground type given in (table 2.10) and (table 2.11). Note that is the earthquake contribute most to the seismic hazard defined for the site have a surface-wave magnitude, M_s, is not greater than 5.5, it is recommended that type 2 spectrum is adopted.

Table 2.10 Recommended values for Type 1 elastic response spectra

Ground type	S	T_B(sec)	T_C(sec)	T_D(sec)
A	1.0	0.15	0.4	2.0
B	1.2	0.15	0.5	2.0
C	1.15	0.20	0.6	2.0
D	1.35	0.20	0.8	2.0
E	1.4	0.15	0.5	2.0

Table 2.11 Recommended values for Type 2 elastic response spectra

Ground type	S	T_B(sec)	T_C(sec)	T_D(sec)
A	1.0	0.05	0.25	1.2
B	1.35	0.05	0.25	1.2
C	1.5	0.10	0.25	1.2
D	1.8	0.10	0.30	1.2
E	1.6	0.05	0.25	1.2

Step 5. Determining the fundamental period of vibration T_1 of the building (Rayleigh method)

For buildings with heights of up to 40 m, the value of T_1 (sec) is given by

$$T_1 = C_t H^{\frac{3}{4}} \qquad (2.35)$$

Where,

- C_t = 0.085 for moment resistant space steel frames, 0.075 for moment resistant space concrete frames and for eccentrically braced steel frames and 0.05 for all other structures.
- H is the height of the building in (m)

Step 6. Distribution of the horizontal seismic forces (applied to two planer models)

The horizontal forces F_i to all storeys

$$F_i = F_b.\frac{s_i m_i}{\sum s_j m_j} \quad \text{2.36}$$

Where;

F_i is the horizontal force acting on the storey

F_b is the seismic base shear

$s_i s_j$ are the displacements of the masses $m_i m_j$ and mj in the fundamental mode shape

$m_i m_j$ = are the storey masses

When the fundamental mode shape is approximated by horizontal displacements increasing linearly along the height, the horizontal forces F_i are given by

$$F_i = F_b.\frac{z_i.m_i}{\sum z_j m_j} \quad \text{2.37}$$

Where $m_i m_j$ are the heights of masses above the level of application of the seismic action.

2.10 Combinations of the seismic actions with other actions.

(a) ASCE CODE

The effects on the structure and its components due to seismic forces are combined with the effects of other loads in accordance with the combinations of load effects given in section 2. Section 2.3.2; Basic Combination:

For structures, components and foundation shall be design so that their design strength equals or exceeds the effects of the factored loads in the following combination:

1.2D + 1.0 E + L + 0.2 S………………………..…………………………..2.38

Where; E = ρQ_E + 0.2 S_{DS} D

In (Section 2.4.1), the load is considered to act in the following combination; whichever produces the most unfavourable effects in the building, foundation or structural members. The effect of one or more loads not acting is considered.

D + H + F + 0.75 (W or 0.7E) + 0.75 L + 0.75 (Lr or S or R)

Where; E = ρQ_E – 0.2 S_{DS} D………...… 2.39

Where:

E is effect of horizontal and vertical induced forces
S_{DS} is design spectral response acceleration at short periods
D is effect of dead load
Q_E is effect of the horizontal seismic (earthquake- induced) forces
ρ is the reliability factor

The most unfavourable effects from both wind and earthquake loads can be considered were appropriate but they need not to be assumed to act simultaneously.

For Special Seismic Load: This special seismic load shall be used to compute E for use in load combination for the equation below

1.2 D + 1.0 E + L + 0.2 S;

Where; E = $\Omega_O Q_E$ + 0.2 S_{DS} D…...................................……………2.40

Where: D is dead load; W is wind load; L is live load; S is snow load; and E is Earthquake Load

(b) EUROCODE

The effects of actions on the structure and its components due to seismic design situation i.e. the design value is determined in accordance with (EN 1990:2002, 6.4.3.4) Combination of action for seismic design situation: (EN 1990:2002, 6.4.3.4)

The general format of effects of action is given as;

$$E_d = E\left\{G_{K,j}; P;\ A_{Ed};\ \Psi_{2,i} Q_{K,i}\right\}\ j \geq 1\ ; i\ \geq 1 \text{...} 2.41a$$

The combination in the bracket {} is expressed as

$$\sum_{j\geq 1} G_{K,j}\ \text{"+"}\ \ P\ \text{"}\ \ A_{Ed}\ \ \text{" + "}\ \sum_{i\geq 1} \Psi_{2,i}\,, Q_{K,i} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 2.41b$$

Table 2.12 Recommended values of Ψ factors for building

Action	Ψ_0	Ψ_1	Ψ_2
Imposed load in buildings, category (see EN 1991-1-1)			
Category A: Domestic, residential areas	0.7	0.5	0.3
Category B: Office areas	0.7	0.5	0.3
Category C: Congregation areas	0.7	0.7	0.6
Category D: Shopping areas	0.7	0.7	0.6
Category E: Storages areas	1.0	0.9	0.8
Category F: Traffic areas (vehicle weight $\leq 30\ kN$)	0.7	0.7	0.6
Category G: Traffic areas (30kN < vehicle weight $\leq 160\ kN$)	0.7	0.5	0.3
Category H: roofs	0	0	0
Snow loads on buildings (see EN 1991-1-3)*			
Finland, Iceland, Norway, Sweden	0.7	0.5	0.2
Remainder of CEN member states, for sites located at altitude H > 1000 m.a.s.1	0.7	0.5	0.2
Remainder of CEN member states, for sites located at altitude H ≤ 1000 m.a.s.1	0.5	0.2	0
Wind load on buildings (see EN 1991-1-4)	0.6	0.2	0
Temperature (non-fire) in building (see EN 1991-1-5)	0.6	0.5	0
NOTE: The Ψ values may be set by the National Annex. & * For countries not mentioned below, see relevant local condition			

Ed is the design value of the effect of action specified in the serviceability criterion, determine in the basis of relevant combination.

The initial effects of the design seismic action is evaluated by taking into account the presence of masses associated with gravity loads

$$\sum G_{k,j} \text{ “+” } \sum \Psi_{E,i} \cdot Q_{k,j} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 2.42$$

$\Psi_{E,i}$ is the combination coefficient for variable action, i see (section 4.2.4)

Table 2.13 Design values for action for use in seismic combination of action

<table>
<tr><th rowspan="2">Design Situation</th><th colspan="2">Permanent actions</th><th rowspan="2">Leading accidental or seismic actions</th><th colspan="2">Accompanying Variable action (**)</th></tr>
<tr><th>Unfavourable</th><th>Favourable</th><th>Main (if any)</th><th>Others</th></tr>
<tr><td>Accidental (*)
(Eq.6.11a/b)</td><td>$G_{kj.sup}$</td><td>$G_{kj.inf}$</td><td>A_d</td><td>Ψ_{11}or
$\Psi_{21} Q_{K1}$</td><td>$\Psi_{2,i} Q_{k,i}$</td></tr>
<tr><td>Seismic
(Eq.6.12a/b)</td><td>$G_{kj.sup}$</td><td>$G_{kj.inf}$</td><td>$\gamma 1 A_{Ek}$ or A_{Ed}</td><td colspan="2">$\Psi_{2,i} Q_{k,i}$</td></tr>
<tr><td colspan="6">(*) In the case of accidental design situations, the main variable action may be taken with it frequent or, as in seismic combinations of actions, its quasi-permanent values. The choice will be in the National annex depending on the accidental action under consideration. See also EN 1991-1-2

(**) Variable actions are those considered in Table A1.1</td></tr>
</table>

2.11 Methods of seismic loads analysis

For seismic performance evaluation, a structural analysis of a building model is simply required to determine force and displacement demands in various components of a structure. Several analytic procedures both linear and nonlinear are available to predict the seismic performance of structures. In this project, equivalent static analysis (ESA) and time history analysis (Nonlinear dynamic analysis) procedures are used.

2.11.1 Equivalent static analysis (ESA)

This method is used to estimate the displacement demands for a structure where a more sophisticated dynamic analysis will not provide additional insight into behaviour. The ESA is well suited for structures or individual frames with well - balanced span and uniformly distributed stiffness where the response can be captured by the predominant translational mode of vibration. In this procedure, seismic load is assumed as an equivalent static horizontal force applied to individual frames. The total applied load is equal to the product of the acceleration response spectrum (ARS) and the weight. The horizontal force is applied at the vertical centre of mass distribution. In this procedure, the horizontal shear force at the base of the structure is estimated as prescribed by ASCE 7 Code. The ESA procedure involves these following steps:

1. Estimate the first mode response period of the building from the design spectra.
2. Use the specific design response spectra to determine that the lateral base shear of the structure is consistent with the level of post-elastic (ductility) response assumed.
3. Distribute the base shear between the various mass levels based on an inverted triangular shear distribution with 90% of the base shear and 10% of the base shear being imposed at the top to allow for higher mode effects
4. Analyse the resulting structure under the assumed distribution of lateral forces and determine the member actions and loads
5. Determine the overall structural response with regard to inter-storey drifts assessed for the elastically responding structure. For post - elastic

deformation, design standards typically modify the elastic deformed shape by the structural ductility to estimate the overall maximum deformation at the roof level.

2.11.2 Nonlinear static (pushover) analysis

This approached also known as "pushover" analysis is a static, nonlinear procedure in which the magnitude of the structural loadings is incrementally increased in accordance with certain predefined pattern. Here, with increase in the magnitude of the loadings, weak links and failure modes of the structure are found. The loading is monotonic with the effects of the cyclic behaviour and load reversals being estimated by modified monotonic force- deformation criteria and with damping approximation. The static pushover analysis is used to evaluate the real strength of the structure and also used as an effective tool for performance base analysis. According to ATC 40, demand and capacity are the key elements for performance based design procedure.

1. Demand is represented by an estimation of the displacements or deformations that the structure is expected to undergo.
2. Capacity is the representation of the structure's ability to resist the seismic demands.

In this case, the structure must have the capacity to resists demand of the earthquake such that the performance of the structure is compatible with the objective of the design.

The pushover analysis is performed by displacement coefficient method/capacity spectrum method (CSM). The CSM is a performance-based seismic analysis technique use for a range of purposes such as:

1. Design verification of new construction of individual buildings
2. Evaluation of an existing structure to identify damage states
3. Rapid evaluation of a large inventory of buildings etc.

The CSM compares the capacity of a structure (in form of pushover curves) with the demands on the structure.

However, the displacement coefficient method is aimed at finding the target displacement which is the maximum displacement that the structure is likely to experience during the design earthquake. This method provides a numerical process for estimating the displacement demand on the structure by simply using a bilinear representation of capacity curves and series of modification coefficients to calculate a target displacement.

2.11.3 Nonlinear dynamic analysis

The nonlinear dynamic analysis utilizes the combination of ground motion records with detailed structural model. In this approached, the detailed structural model is subjected to a ground motion record produces estimates of the components deformation for each degree of freedom in the model and the model responses are combined using schemes such as the square-root-of-squares. Nonlinear dynamic analysis is capable of producing a relatively low uncertainty. Calculated response in this approach can be sensitive to the characteristic of the individual ground motion used as inputs; hence for more reliable estimation of the probabilistic distribution of the structural responses, several analyses by using different ground motion is needed.

Modelling of inertia mass and gravity loads: The inertial mass is the expected mass, including the self-weight plus some allowance based on similar assumptions used in the determination of seismic masses for design per ASCE 7 or other codes. It's usually adequate to lump the masses at the floor levels and to include the inertial effects in the two horizontal directions with rotation inclusive along the vertical building axis. For long span framing, the vertical inertial effects i.e. vertical mass and ground motion components should be modelled where the vertical period of vibration is in the range that may be excited by the vertical component of the earthquake ground motions. Where member are sensitive to the vertical loads, the influence of the code-specified vertical earthquake load e.g. the *E* factor in ASCE 7 should be accounted for in the force demands calculation.

Gravity Loads is included in the dynamic analysis so as to account for their effects on; force and deformation demands in structural components and large displacement P- delta effects. The inclusion of the gravity loads requires two steps (non-proportional loading) analysis in a way that the gravity loads are applied first and then held constant while the earthquake ground motions is applied.

2.11.4 Discussion of different analysis method

In the past, most studies were focused on the use of equivalent static analysis which defines a series of forces acting on a structure to represent the effect of earthquake ground motion, typically define a seismic design response spectrum. However, this approach is much more applicable only to regular structure which satisfy the condition given; a structure that have orthogonal layout and shall not be too flexible or in plane, the building shall not be unbalanced in its distribution of mass or stiffness. So due to need for approximation of adequate information on seismic demands imposed by design ground motion on the structure, pushover analysis is much preferred as it gives expected performance of structural systems by estimating its strength and demands in design earthquakes. Also nonlinear dynamic analysis if properly implemented, accurate calculation of structural response to strong ground motion can be achieved.

Most research papers on single-storey asymmetric buildings models have been focusing on the inelastic behaviour of resisting elements to simplify analysis and to facilitate parametric studies (De Stefano et al. 1993, Myslimaj and Tso 2002, De la Llera and Chopra 1994b, De Stafano and Pintucchi 2004), the effect of coupling between lateral and torsional motion is frequently studied in terms of elements ductility demand. It should be clearly noted that for regular building structures, seismic performance of plan asymmetric framed structures mainly relies on a ductile response.

Gulkan and Sozen (1974) noted that most of the time displacement would be significantly smaller than the maximum response under earthquake loading. Thus the equivalent damping proposed by Rosenbluth and Herrera (1964) would result in an overestimation of equivalent viscous damping that the response would be underestimated. Gulkan and Sozen (1974) developed an empirical equation for equivalent damping ratio using secant stiffness Takeda hysteretic model (1970) and the results obtained from experiments made on single story, single bay frames supported the proposed procedure.

Looking back to the work of Gulkan and Sozen (1974) where a single degree of freedom of system is derived to represent equivalently the multi-degree of freedom structure. The load displacement curve of this substitute is evaluated by either finite element analysis or hand calculation to obtain the initial and post yield stiffness, the yield strength and the ultimate strength. Simplified inelastic analysis procedures for multi-degree of freedom systems have also been proposed by Saudi and Sozan (1981) and Fajfar and Fischingr (1988), their work was focused on the use of multi-degree of freedom inelastic analysis of complex structures, which is a recent development.

Besides, Fajfar and Fischinger (1987) also proposed the N2 method as a simple nonlinear procedure for seismic damage analysis of reinforced concrete buildings. The method uses response spectrum approach and nonlinear static analysis. This method was applied to three 7-story buildings Fajfar and Gaspersic (1996). The capacity curve of a MDOF system was converted to that of a SDOF and a global demand was obtained. A damage model which includes cumulative damage was determined at global demand. The method yields reasonably accurate results provided that the structure vibrates predominantly in the first mode.

Newmark and Hall (1982) and Miranda (2000) proposed procedures based on displacement modification factors in which the maximum inelastic displacement demand of MDOF system is estimated by applying certain displacement modification factors to maximum deformation of equivalent elastic SDOF system having the same lateral stiffness and damping coefficient as that of MDOF system. Similarly, Displacement Coefficient Method described in FEMA-356 (2000) is a non-iterative approximate procedure based on displacement modification factors. The expected maximum inelastic displacement of nonlinear MDOF system is obtained by modifying the elastic spectral displacement of an equivalent SDOF system with a series of coefficients.

More so, Chopra and Goel (1999-2002) have proposed an improved capacity-demand diagram method that uses constant ductility demand spectrum to estimate seismic deformation of inelastic SDOF systems.

Last but not the least, Stathopoulos and Anagnostopoulos (2005) investigated the inelastic earthquake response of eccentric multi-storey RC frame building by means of 3 and 5- storey models subjected to bi-directional motions. In the building designed, both the EC8 provisions and the UBC-97 codes were considered. However, the result suggested that frames at the flexible side experience increased inelastic deformation, while those at the stiff side decreased deformations with respect to their symmetric counterparts. Despite emphasizing that more studies are needed, they came up with final conclusion that such uneven distribution of demands is certainly an indication that current codes need re-examination. More so, these findings contradict those obtained from a single-storey model, which are considered inadequate for predicting the main qualitative features of the inelastic response for multi-storey structures.

CHAPTER THREE

METHODOLOGY

3.1 Introduction

The methodology includes modelling and analysis of the canopy structure in STAAD Pro software. In the analysis, methods such as equivalent static analysis and nonlinear dynamic analysis are applied to determine the canopy responses to the seismic loadings. In the design stage, capacity design principle shall be applied to design the structure to have a suitable ductile yielding mechanism under nonlinear lateral deformation. The nonlinear procedure includes performing a nonlinear dynamic analysis of the structure with appropriate lateral load pattern. However, linear analysis may be used to estimate the require strength of the yielding actions and hence, predicting the important response parameters i.e. maximum displacement demands and capacity of the structure shall be demonstrated by using distinct levels of seismic ground motion. The methodology to achieve objectives in section 1.3 is discussed in the sections outlined below:

3.2 Software use

The software, STAAD Pro will be used in modelling and analysis of the structure. STAAD Pro is a structural analysis and design software which supports steel, concrete, and timber design codes. The software can make use of various forms of analysis from first- order static analysis, second-order P- Δ Analysis, geometric nonlinear analysis to various form of dynamic analysis from model extraction etc.

3.3 Modelling and analysis of the canopy structure

The structure is modelled and analysed for both nonlinear static analysis and nonlinear dynamic analysis. The modelling of structure in STAAD Pro software is done with regards to the used of standard provision and codes. Most structures are designed to have adequate strength and stiffness to resist the applied loads due to gravity and wind loading. The model contains beams and columns must ideally represent the complete 3D characteristic of the building including geometry, stiffness of various member, supports, load distribution, mass distribution. Beams and columns are modelled by frame elements and plinth beams should be modelled as beam, however slabs are not usually modelled. Support type to be used is decided by the degree of fixity provided by the foundation. The principal loadings are mainly self-weight of structure and its occupancy. This simply refers to the “dead” loads which comprises of the weight of the supporting structure together with the weight of the finishes. Live loads, is the load that the floor or roof is likely to sustain during its life and will depends on the use. To achieve Sub-objectives in [section 1.3], the following equivalent static analysis and nonlinear dynamic analysis steps are conducted in [section 3.4 and 3.5].

3.4 Equivalent static analysis (ESA)

A static analysis simply calculate the effects of steady loading conditions on a canopy structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. This method however includes steady inertia loads such as gravity loads and time-varying loads that can be approximated as static equivalent loads i.e. Static equivalent wind and seismic loads commonly define in many building codes.

3.4.1 Loads in static analysis

Equivalent static analysis simply determines the displacements, stresses, strains and forces in the structure or components caused by loads that do not induce significant inertia and damping effects. In this analysis, the loads and the structure’s response are assumed to vary slowly with respect to time. The primary type of the loadings applied in a static analysis of the canopy includes dead loads, live loads, wind loads and seismic loads.

3.4.2 Steps in a static analysis of canopy structure.

The equivalent static analysis method for seismic load analysis has three main steps which include:

1. Building of the model i.e. canopy structure

To build the model, specify the job name and analysis title and then use the pre-processor to define the elements type, element constants, material properties, and the model geometry.

2. Applying the loads and perform analysis

Define the analysis type and options, apply loads, specify load step options, and start the finite element analysis.

3. Review the results.

The results from equivalent static analysis are written to the structural results file. They consist of the following data:

- Primary data: Nodal displacements (UX, UY, UZ, ROTX,ROTX, ROTZ)
- Derived data: Nodal and elements stresses, Nodal and element strains, element forces, Nodal reactions forces etc.

3.5 Nonlinear Dynamic Analysis (NDA):

This method of analysis utilizes the combination of ground motion records with detailed structural model. In this approach, the detailed canopy structural model is subjected to seismic ground acceleration obtained from the Malaysian Meteorological Department (MMD). The ground motion of magnitude 9.1 (source: USGS) located off west coast of Northern Sumatra at a latitude/longitudinal: 3.2^0N 95.9^0E was recorded on 26/12/2004. When the model is subjected to such a ground motion, estimates of the component deformation produced for each degree of freedom in the model responses are combined using schemes such as square-root-of-squares.

3.5.1 Modelling of canopy structure

Model of the structure using STAAD Pro is constructed to represent the spatial distribution of the masses throughout the structure including self-weight of the building as per EN 8. In this experiment, the structure is assumed to have fixed base with design ground motion to occur along any horizontal direction of the structure.

3.5.2 STAAD Pro 3D model of a canopy

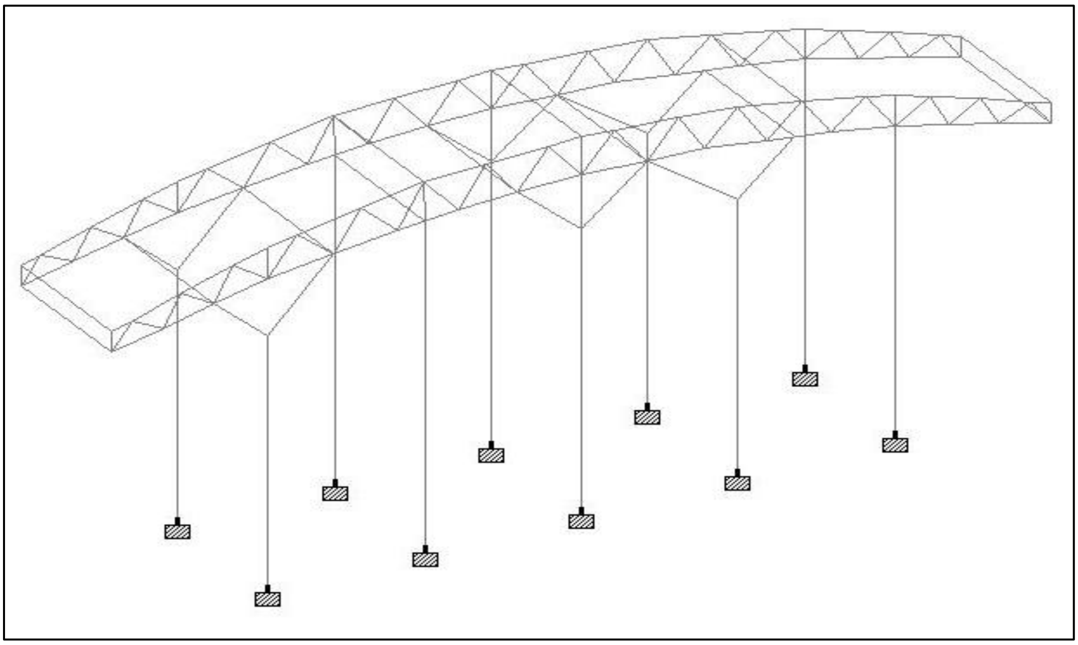

Figure 3.1 STAAD Pro 3D model of a canopy

3.5.3 Input seismic ground motion.

In this project, the ground motion selected is an actual recorded data from the past earthquakes and it reflects the characteristics of the dominant earthquake in Sumatra, Indonesia felt in Penang, Malaysia. According to (Baker and Cornell 2006) from their study emphasize that the shape of the ground motion spectra is an important factor in choosing and scaling the ground motion for building assessment and design. This data was taken as the input ground motion for the analysis.

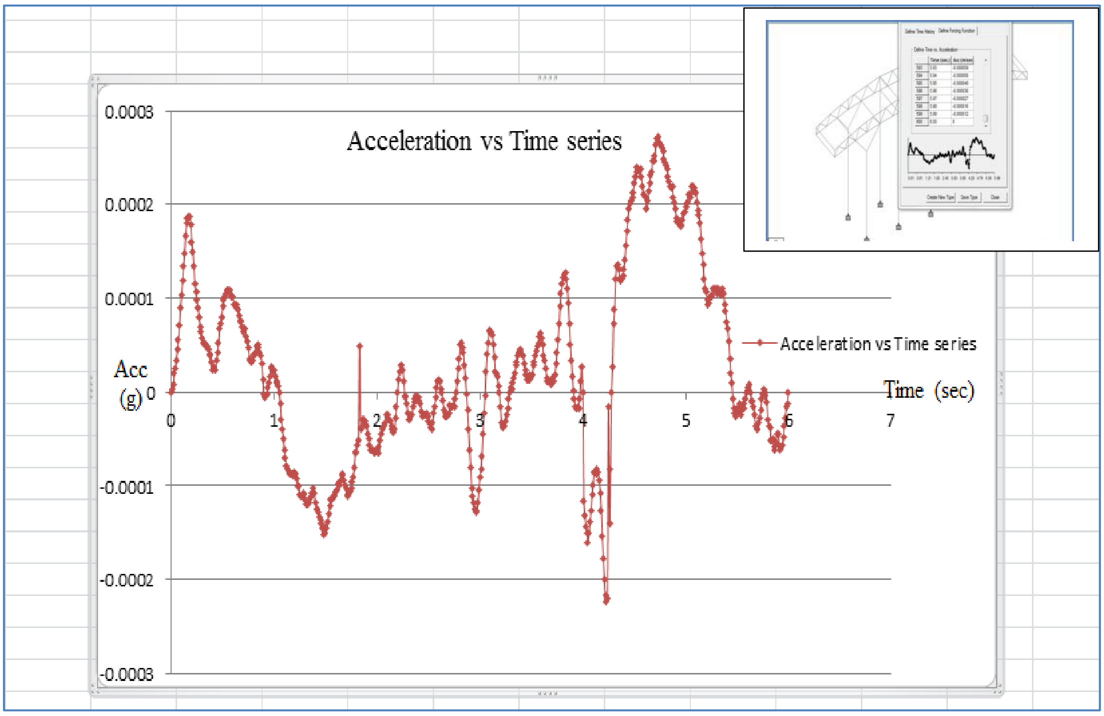

Figure 3.2 Acceleration vs. time series

3.5.4 Damping effects.

Damping is a property of material which influences dynamic response. In time history analysis, equivalent viscous damping results from reduction in vibrations through energy dissipation other than that which is calculated directly by nonlinear hysteresis in the model elements. The value of viscous damping to be applied consistently is 3% of critical damping. However, the equivalent damping is proportional to the mass and stiffness.

3.5.5 Steps in a Time History Analysis of canopy structure.

To perform time history analysis, the following process is initiated.

1. Create the model and assign support condition to restrained joints
2. Select Define > Functions > Time History to define a time-history function which characterize load variation over time.
3. Assign load conditions to the model through Assign > joint loads or frame loads.
4. Define either a model or direct-integration time-history load case through Define > load cases

5. To customize load application, enter, on the Load Case Data from under Load Applied, the following fields:
    - Load Type: Select the load type to be applied at the structure’s base, (e.g. Force or Acceleration)
    - Load Name. Select the load pattern to be applied
    - Function. Select the time function which characterizes load variation in time.ie (Time vs. acceleration, harmonic function, from external file)

The masses that constitute the mass matrix of the structure are specified through the self-weight and joint load commands. The programme will extract the lumped masses from the weights.

During the analysis, the programme calculates joint displacements for very time step. The absolute maximum value of the displacement for every joint is then extracted from this joint displacement history and printed.

3.6 Project processes flow

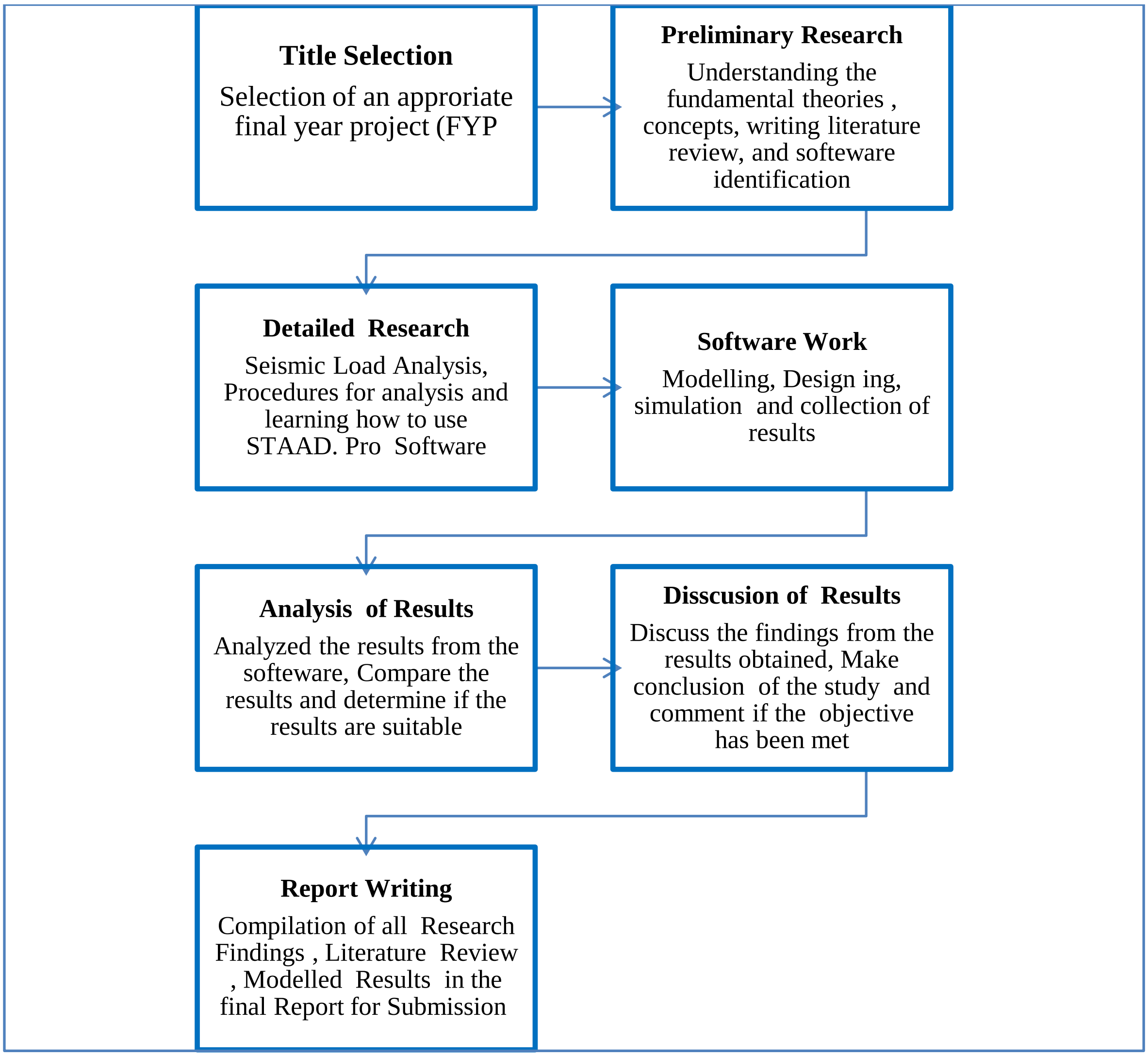

Figure 3.3 Project process flow

3.7 Key milestone and project planning

Activities	Final Year Project I (FYP 1)														Final Year Project II (FYP 2)													
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Briefing, Topic selection and Confirmation																												
Abstract, Literature-review & Methodology																												
Research Continues, Extended Proposal submission & Proposal defence (Presentation)																												
Draft Report (Interim Report) & Interim Report Submission																												
Canopy modelling, Data Calculation & design in STAAD. Pro software.																												
Performing STAAD. Pro simulation work, Check and analysis of Output (Results).																												
Submission of progress report, results discussion & Conclusion work and Pre-EDX																												
Submission of Draft report, dissertation (soft bound) and technical paper																												
Compilation of work, presentations and submission of project dissertation																												
Milestone	Final Year Project I (FYP-1)														Final Year Project II (FYP-2)													
	1	2	3	4	5	6	7	8	9	10	11	12	13	14	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Completion of Abstract, Literature Review and Methodology work and report submission																												
Completion of Modelling, Data calculation, and Design of structure in STAAD.Pro																												
Completion of simulation work in STAAD. Pro, Analysis and Discussion of results																												
Comparison of results of ESA & NDA and Report Submission																												

Figure 3.4 Gantt chart for final year project (FYP 1) and (FYP 2) Implementation and Milestone

CHAPTER FOUR

RESULTS AND DISCUSSION

4.1 Description of the typical Canopy structure.

The canopy structure of 36m length and spacing 6m is chosen for the analysis by using STAAD Pro program. The height of the structure is designed to be approximately 17m with weight of the canopy structure estimated to be 465kN. Shown in (figure 4.1) and (figure 4.2) is the canopy structure designed details.

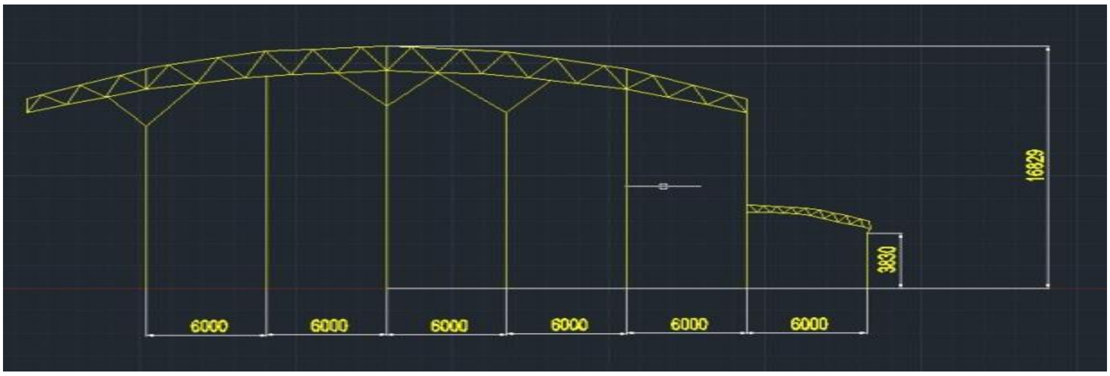

Figure 4.1 AutoCAD model of a canopy

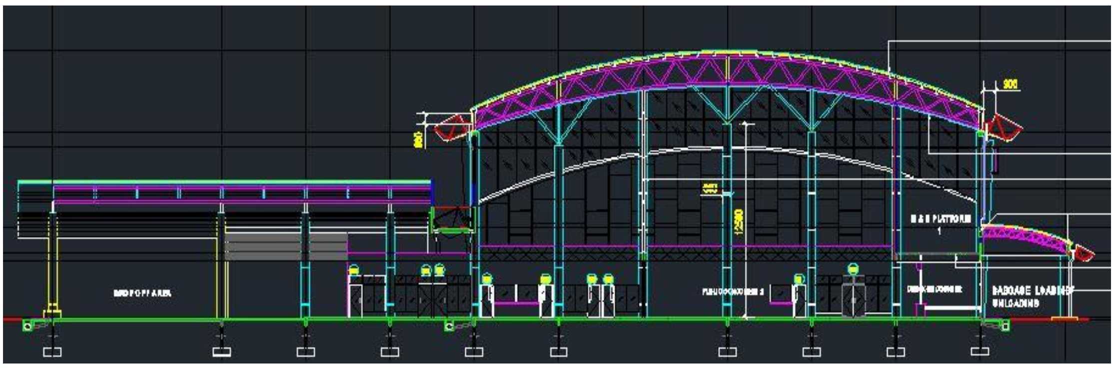

Figure 4.2 Canopy cross-section

4.2 Design Loadings (Eurocode)

Permanent Actions

S/No	Structural materials (self-weight)	Value (kN/m^2)
1	Roof sheeting & insulation	0.40
2	Purlin	0.15
3	Lighting	0.10
4	Ceiling	0.15
	Total Load	**0.80**

5	Glass-20 mm thick	0.50
6	Aluminium frame	0.10
	Total Load	**0.60**

Variable actions

Assume category H, roof not accessible except for normal maintenance and repair work. UDL selection range (0.0 - 1.0 kN/m^2)

S/NO	Type	Value (kN/m^2)
1	Roof s	0.40
	Total Load	**0.40**

Wind actions

S/NO	Type	Value (KN/m^2)
1	Wind loads (wind speed, zone 1: 33.5m/s)	1.264
	Total Load	**1.264**

4.3 Structural details: (Determination of loadings on the building envelope).

- Total length of structure = 36 m
- Spacing = 6 m
- Height (max) = 16.879 m ≈ 17 m

The safety factors for action on building structures for persistent and transient design situation action based on the Eurocode (**EN 1991**)

Actions	Safety factor
Permanent action	1.35
Variable actions	1.50
Wind actions	1.00
Seismic action	1.00

4.4 Materials property and constant

The canopy structure is entirely designed of steel. Steel provides safe, reliable and ductile structures. During earthquake, steel has the greatest performance. The steel material is good at dissipating energy due to:

1. Ductility of steel material
2. Durable and long term-tem consistent performance
3. Energy absorption capacity i.e. Steel bends without breaking
4. Steel are lighter hence less inertia
5. Many possible mechanism in steel elements and in connection
6. Reliable geometrical properties
7. Guaranteed material strength due to controlled production
8. Design and construction made by professional

Structural Steels: Grade S235, S275, S355, S450 steels, conforming to EN 10025-2: 2004. (Geocentrix, 2012)

Wight density	76.98 kN/m3
E (Young's modulus)	210 GPa
v (Poisson's ratio)	0.3

4.5 Determination of wind load

Step 1: Determination of basic wind velocity.

From the **MS 1553:2002**, the value 33.5 m/s zone 1 is selected with reference based on 50 years return period.

$V_b = C_{dir}.C_{season}.V_{b,0}$ .... [**EN 1991-1-4 Clause 4.2**]

Where;

C_{dir} is directional factor (recommended 1.0)

C_{season} is seasonal factor (recommended 1.0)

$V_{b,0}$ is fundamental value of the basic wind velocity

V_b is basic wind velocity (10m above ground, cat.II)

Terrain category II (z_o = 0.05 m & z > z_{min}) is area with low vegetation such as grass and isolated obstacles (trees and building) with respects at least 20 obstacle heights. [**EN 1991-1-4 Clause 4.3.2. Table 4.1**]

$V_{b,0} = V_b.C_{prob}$ (C_{prob} is probability factor)

$$C_{prob} = \left(\frac{1-k\,.In\left(-In\,(1-p)\right)}{1-k.\;In\left(-In\,(0.98)\right)}\right)^n$$

(Recommended values k = 0.2 and n= 0.5)

$$C_{prob} = \left(\frac{1-0.2\,.In(-In\,(1-0.02))}{1-0.2.In\,(-In\,(0.98))}\right)^{0.5}$$

C_{prob} = 1, [the annual probability of exceedance p = 0.02 (return period, R= $\frac{1}{p}$ = 50 years)]

$V_{b,0} = V_b.C_{prob}$ = 33.5 x 1.0 = 33.5 m/s

$V_b = C_{dir}.C_{season}.V_{b,0}$ = 1.0 x 1.0 x 33.5 = 33.5 m/s

Step 2: Mean wind Velocity: V_m (z) at height z

$V_m (z) = C_r (z) C_o (z) V_b$ ……………………………….……… [**EN1991-1-4. Clause 4.3.1**]

C_o (z) is orography factor (taken as 1.0)

C_r (z) is the roughness factor: .... [**EN1991-1-4. Clause 4.3.1**]

$C_r (z) = k_r.\text{In} (\frac{z}{z_o})$ for $z_{min} \leq z \leq z_{max}$ and $C_r (Z) = C_r (Z_{max})$ for $z \leq z_{max}$

$K_r = 0.19 (\frac{z_0}{z_{0,II}})^{0.07}$

$z_{0,II}$ = 0.05 (Terrain category II, table 4.1)

z_{min} = minimum height (Table 4.1) and z_{max} = is taken as 200m

$K_r = 0.19 \left(\frac{0.05}{200}\right)^{0.07} = 0.106 \approx 0.11$

$C_r (z) = 0.11 \text{ In } \left(\frac{17}{0.05}\right) = 0.64$

V_m (z) = 0.64 x 1.0 x 33.5 = 21.44 m/s

Terrain Orography: Effect neglected when the average slope of the upwind terrain is less than 3^0

Wind turbulence: I_v (z),

$\delta_v = k_r. V_b. k_1$

k_r = Terrain factor, k_1 = turbulent factor (k_1= 1.0)

$I_v(z) = \frac{\delta_v}{V_{m(z)}} = \frac{k_1}{C_{0(z)}\, In\,(\frac{z}{z_0})}$ for $z_{min} \leq z \leq z_{max}$……..… **[EN 1991-1-4 Clause 4.4]**

$I_v(z) = I_v(z_{min})$ for $z < z_{min}$; z_0 is the roughness length (Table 4.1)

Step 3: Peak velocity pressure: $q_b(z)$ at height z **[EN 1991-1-4 Clause 4.5]**

$q_p(z) = [1+ 7.I_v(z)].\frac{1}{2}\rho V_m^2(z) = C_e(z).q_b$

C_e = exposure factor

$C_e(z) = \frac{q_{p(z)}}{q_b}$ (for flat terrain $C_0(z) = 1.0$

$q_p(z) = [1+7.\left(\frac{k1}{C_{0(Z)}\ In\frac{Z}{Z_0}}\right)].\frac{1}{2}\rho V_m^2(z) = [1 + 7\left(\frac{1.0}{1.0\ In\frac{17}{0.05}}\right)]\text{x } 0.5 \text{ x } 1.25 \text{ x } (21.44)^2$

$q_p(z) = 2.201 \text{ x } 287.296$

$$\mathbf{q_p(z) = 632.34\ N/m^2\ (0.632\ kN/m^2)}$$

Using $q_p(z) = Ce(z).\ q_b = \frac{qp\ (z)}{qb}.\ q_b = 632.34\ N/m^2$

Step 4: Basic velocity Pressure:

$q_b = \frac{1}{2}\rho_{air}.\ V_b^2$…………………………………………..… **[EN 1991-1-4 Clause 4.5]**

Where: $\rho_{air} = 1.25\ kg/m^3$ (density of air)

$$\mathbf{q_b = 0.5 \text{ x } 1.25 \text{ x } 33.5^2 = 701.41\ N/m^2\ (0.701\ kN/m^2)}$$

Wind Action: The wind action taken into account is both external and internal wind pressure on the structure.

Step 5: Wind pressure on external surface (W_e) of structure.

$W_e = q_p(z).\ C_{pe}$……………………………………… **[EN 1991-1-4 Clause 5.2(1)]**

Where:

q_p (ze) - Peak velocity pressure

z_e is reference height for external pressure

C_{pe} is pressure coefficient for external pressure (depending on the size of the loaded area A = $C_{pe, 10}$ because the loaded area A for the structure is larger than 10 m^2)

Step 6: Wind pressure on internal surface (W_i) of structure.

$W_i = q_p$ (zi). C_{pi}………………………………………. **[EN 1991-1-4 Clause 5.2(1)]**

Where:

q_p (zi) - Peak velocity pressure

z_i is reference height for internal pressure

C_{pi} is pressure coefficient for internal pressure

Wind actions:

The wind loading per unit length (w in kN/m) for an internal frame are calculated using the influence width (spacing) s = 6.0 m

Wind load (W) = ($C_{pe} + C_{pi}$) q_p S

Note: Consider the external and internal pressures to act at the same time and the worst combination of external and internal pressure for every combination of possible opening and other leakage paths.

Canopy wind force coefficient

This coefficient is for a duo pitch canopy structure. The slope roof is assumed to be 10^0. The relevant force coefficient are taken as -1.3 for upward wind loading and + 0.7 for downward wind loading

The downward wind force is: **0.632 x 0.7 = 0.442 kN/m**

The upward wind force is: **0.632 x (-1.3) = - 0.822 kN/m**

Total wind force = (0.442 + 0.822) = 1.264 kN/m

4.6 Determination of seismic loads

The procedure of **Eurocode (EN 1998)** seismic load determination is based on the equivalent Static Analysis (ESA).This method is used to estimate the displacement demands for a structure where a more sophisticated dynamic analysis will not provide additional insight into behaviour.

In this method the inertia effects of the design seismic action is evaluated by taking into account the presence of the masses associated with gravity loads**. [EN1998-3.2.4]**

$\sum G_{k,j}$ "+" $\sum \Psi_{E,i} \,.\, Q_{k,j}$, where $\Psi_{E,i}$ is the combination coefficient for variable action.

- Weight of steel roofing (assuming 75 mm diameter), $\sum W_{roof}$ = 245 kN
- Weight of column (assuming 75 mm diameter), $\sum W_{Column}$ = 139 kN

Estimated total weight of the building

Weight (W) = $\sum W_{roof}$ + $\sum W_{column}$ + 1.5 Variable action

W = 245 kN **+ 139** kN **+ 1.5 x (54) = 465 kN**

Base shear force:

Use the given equation to determine the base shear force F_b for each horizontal direction.

$F_b = S_d(T_1).m.\lambda$... **[EN 1998-4.5]**

Where; $S_d(T_1)$ is the ordinate of the design spectrum at period T_1

T_1 is the fundamental period of vibration of the building

m is the total mass of the building computed according to 3.2.4(2)

λ is the correction factor, the value of λ = 0.85 if $T_1 \leq 2T_c$ or a building having more than 2 storeys λ = 1.0

To determine the fundamental period of vibration T_1 of the building (Rayleigh method) for buildings with heights of up to 40 m, the value of T_1 (sec) is given by

$T_1 = C_t H^{\frac{3}{4}}$.. **[EN 1998, eqn 4.6]**

Where,

C_t = 0.085 for moment resistant space steel frames and $H = 17\ m$ is the height of the building.

$$\mathbf{T = 0.085 \times 17^{\frac{3}{4}} = 0.712\ sec}$$

Using type 1 elastic response spectra class C. Data are given in **[EN 1998-Table 3.2]**

Table 4.1 Recommended values for Type 1 elastic response spectra

Ground type	S	T_B(sec)	T_C(sec)	T_D(sec)
A	1.0	0.15	0.4	2.0
B	1.2	0.15	0.5	2.0
C	**1.15**	**0.20**	**0.6**	**2.0**
D	1.35	0.20	0.8	2.0
E	1.4	0.15	0.5	2.0

Using; $T_C \leq T \leq T_D : S_d(T) \begin{cases} = a_g \, .S.\frac{2.5}{q}.\left[\frac{T_C}{T}\right] \\ \geq \beta \, . a_g \end{cases}$...........Eqn 3.15 **[EN1998- 3.2.2.5]**

Where

$a_g = \gamma_1 \; x \; a_{gR}$ = 1 x 0.25g = 0.25g

β is the lower bound factor for the horizontal design spectrum, $\beta = 0.2$

q is the behaviour factor from the expression; q = q_o x k_w $\geq$ 1.5 **[EN 1998 -5.2.2.2]**

From EN 1998-Table 5.1 is q_0 = 3.0 x $\frac{\alpha_u}{\alpha_1}$ and from EN 1998-5.2.2.2(5) which the ratio $\frac{\alpha_u}{\alpha_1}$ is 1.3 multi-storey or frame equivalent dual structure q_0 = 3 x 1.3=3.9

k_w is the factor reflecting the prevailing failure mode in structural system with wall. k_w is calculated from EN 1998-5.2.2.2(11) takes it as 1 for frame and frame equivalent dual systems.

$$S_d(T_1) = a_g \, .S.\frac{2.5}{q}.\left[\frac{T_C}{T}\right]$$

$S_d(T_1) =$ 0.25x 1.15 x $\left(\frac{2.5}{3.9}\right)$ x $\left(\frac{0.6}{o.712}\right)$ = 0.155g and $\geq$ $\beta \, . a_g$ = 0.2 x 0.25 = 0.05

Using; $F_b = S_d(T_1).m.\lambda$... **[EN 1998-4.5]**

F_b = 0.155g x $\frac{W}{g}$ x 0.85 = 0.13175 W

$\boldsymbol{F_b}$= 0.13175 x 465 = 61.264 kN

4.7 STAAD Pro 3D model of the canopy

Figure (4.3) shows a 3D model in STAAD Pro 2004 program. The model is analysed for permanent load, variable load, wind load and seismic load acting on the structure. The software program will calculate the bending moments, shear and axial forces of the structural elements and as well determine the lateral displacement due to the code forces.

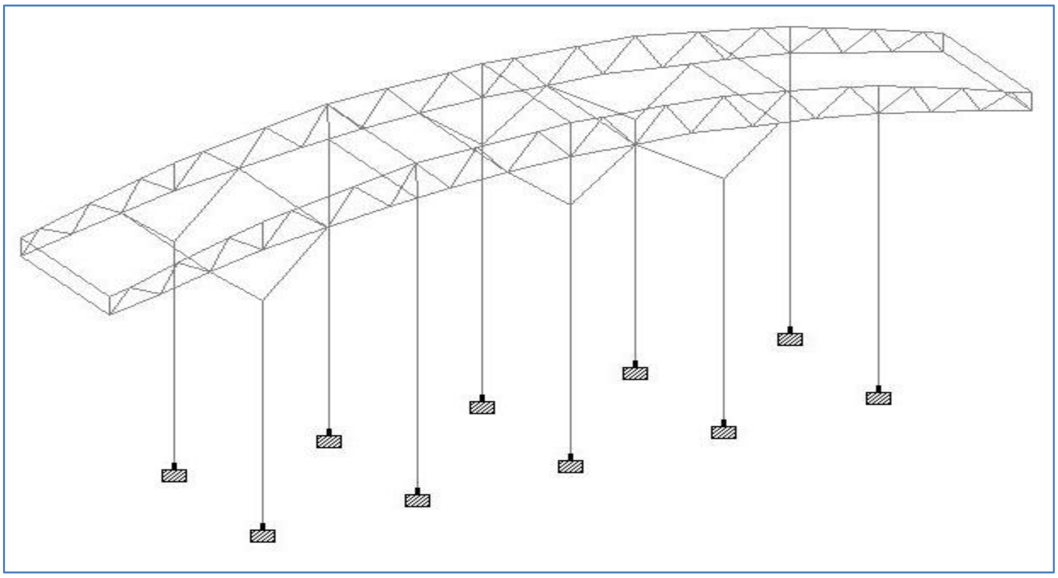

Figure 4.3 STAAD Pro 3D model of the canopy

4.8 STAAD Pro simulation

In STAAD Pro 2004, regardless of the structure being analysed, the following are fundamental steps and STAAD Pro command keywords shown in the brackets:

1. Define whether the problem is 2D or 3D (STAAD PLANE or SPACE)
2. Define the length and force units (UNITS)
3. Define the nodes and their locations (JOINT COORDINATES)
4. Define the member and their nodes (MEMBER INCIDENCES)
5. Define the section properties of the members, Ix, etc. (MEMBER PROPERTY)
6. Define the mechanical properties of the members such as the Young's modulus, density, etc. (CONSTANTS)
7. Define the support conditions (SUPPORTS)
8. Define the load cases (LOAD)
9. Define the loads of each load case as member loads, joint loads, (or code loads)(MEMBER LOAD or JOINT LOAD)
10. Define the load combinations (LOAD COMB)
11. Analyse the structure (PERFORM ANALYSIS)
12. Define the output format (PRINT)
13. Finish the run (FINISH)

4.9 STAAD Pro 2004 simulation results for given load combination 5 & 6

LOAD CASES

Type	L/C	Name
Primary	1	DEAD LOAD
Primary	2	LIVE LOAD
Primary	3	WIND LOAD
Primary	4	SEISMIC LOAD
Combination	5	DEAD LOAD + LIVE LOAD + WIND LOAD
Combination	6	DEAD LOAD + LIVE LOAD + SEISMIC LOAD

EQUIVALENT STATIC ANALYSIS (ESA) METHOD

(1) Load combination 5 = (Dead Load + Live Load + Wind Load)

These are the displacement, beam stresses, bending and shear shapes of the canopy structure due to load combination 5(including wind effect)

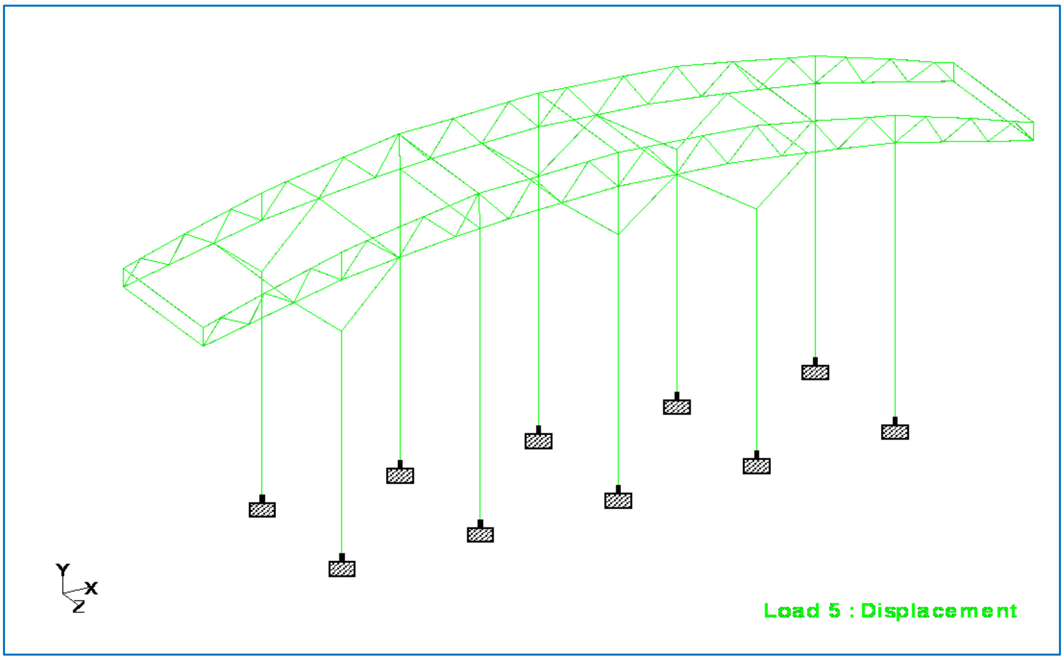

Figure 4.4 Structure nodal displacements due to wind effect

The structure nodal displacement as shown in (figure 4.4) is due to the wind load acting on the structure. The wind effect caused a maximum displacement of 8 mm as shown in table 4.2 or refer to **(Appendix 4.2)** for details.

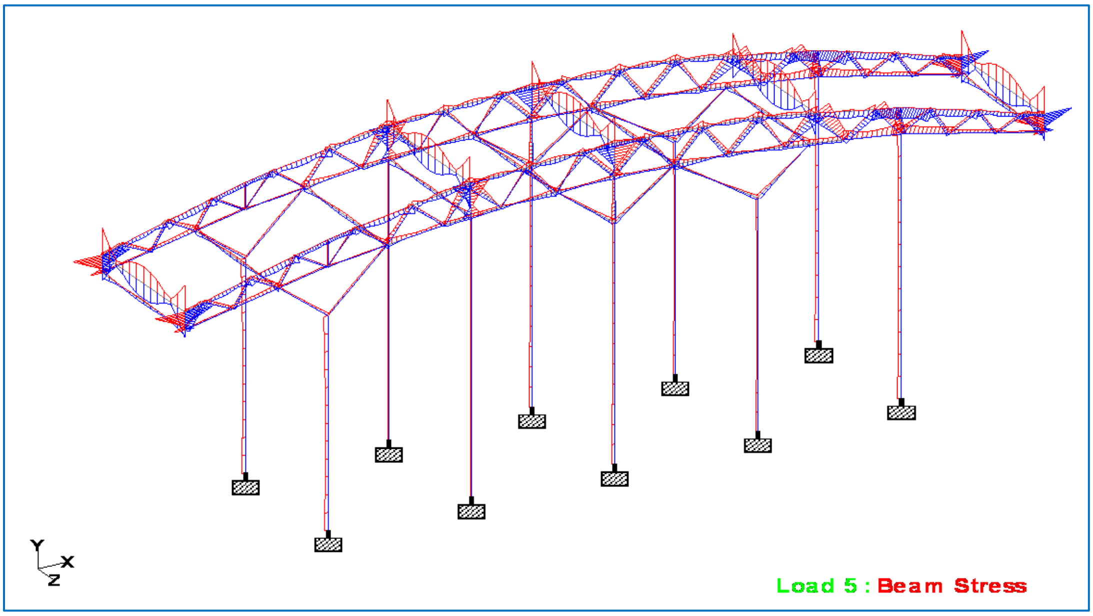

Figure 4.5 Structure beam stresses due to wind effect

The beam stresses in (figure 4.5) on the structural members are as a result of the applied wind load. Beams often developed normal stresses when subjected to external loads acting perpendicularly. The wind load acting on the beams generated bending moments as shown in figure 4.6 along Y and Z axis.

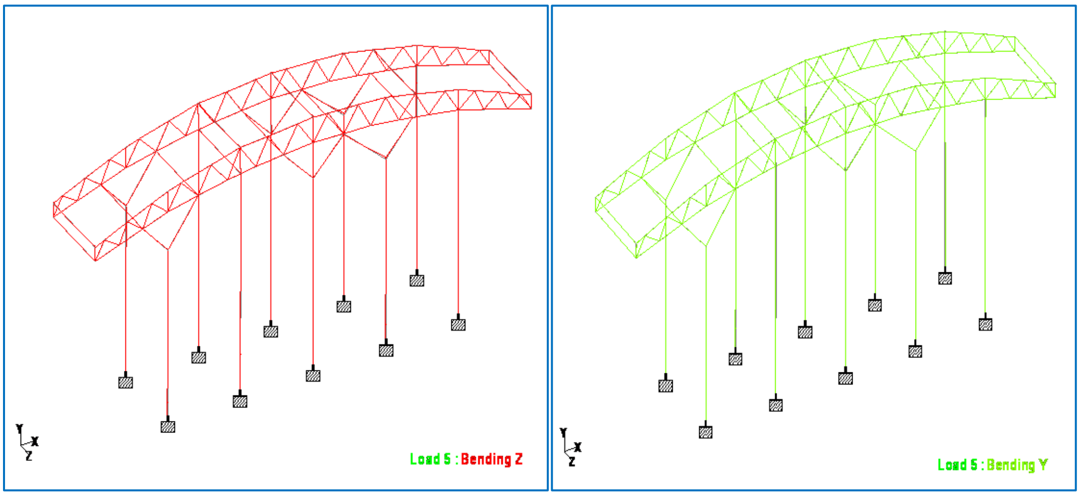

Figure 4.6 Bending moment acting along Z and Y axes due to wind effects

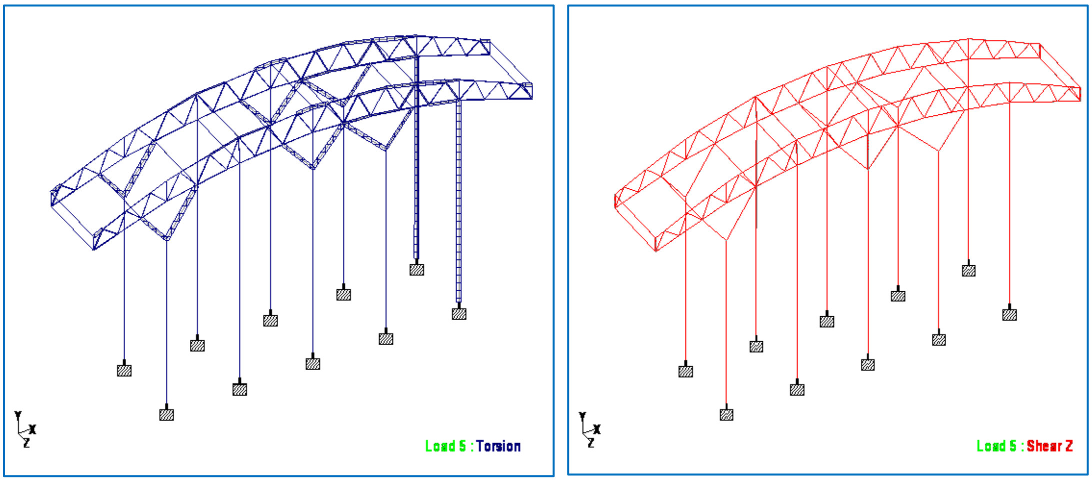

Figure 4.7 Torsion and shear force along X and Z axis due to wind effect

Figure 4.7 indicates the structural shear forces due to the wind effect. The shear force acting on the structure is along the Z and Y axis. The detailed results of the shears forces acting on the structure is shown in (**Appendix 4-3)**

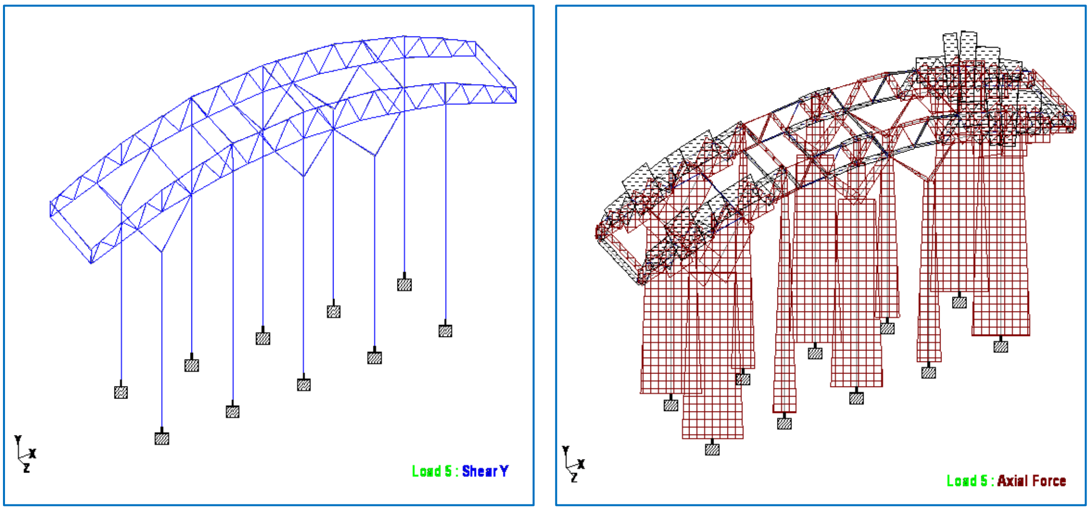

Figure 4.8 Shear and axial force acting along Y and X axis due to wind effects

Figure 4.8 indicates the torsional moment and the axial force acting on the structure due to the wind effect. The maximum torsion acting on the structure is 0.008 rad as shown in **(Appendix 4-2).** The torsional effect is small, it is indicative that the structure wall framing is symmetrical with respect to the centre of mass.

(2) Load combination 6 = (Dead Load + Live Load + Seismic Load)

These are the displacement, beam stresses, bending and shear shapes of the canopy structure due to load combination 6 (including seismic effect)

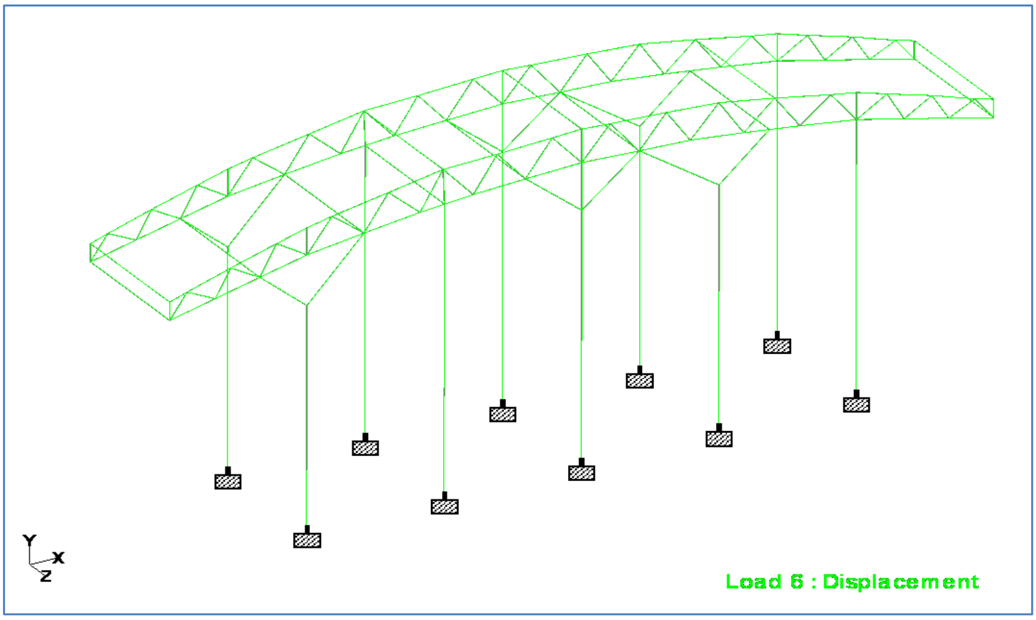

Figure 4.9 Structure nodal displacement due to seismic effect

The figure 4.9 indicates the structural nodal displacement as a result of the seismic effects on the structure. The maximum displacement due to the seismic load recorded is 25 mm as shown in table 4.2 or refer to **(Appendix 4.2)** for details.

Table 4.2 Summary of maximum Nodal Displacement

	Node	L/C	Horizontal X m	Vertical Y m	Horizontal Z m	Resultant m	Rotational rX rad	rY rad	rZ rad
Max X	37	6 DEAD LOAD + LIVE LOAD + SEISMIC LOAD	**0.025**	-0.002	0.000	0.025	0.000	0.000	-0.001
Min X	52	1 DEAD LOAD	**-0.001**	-0.003	-0.000	0.003	-0.000	-0.000	-0.001
Max Y	1	4 SEISMIC LOAD	0.023	**0.001**	-0.000	0.023	-0.000	-0.000	-0.000
Min Y	130	6 DEAD LOAD + LIVE LOAD + SEISMIC LOAD	0.022	**-0.005**	-0.000	0.023	-0.003	0.001	-0.001
Max Z	116	5 DEAD LOAD + LIVE LOAD + WIND LOAD	0.001	-0.000	**0.008**	0.008	-0.003	0.002	-0.000
Min Z	39	5 DEAD LOAD + LIVE LOAD + WIND LOAD	0.001	-0.000	**-0.008**	0.008	0.003	-0.002	-0.000
Max rX	38	5 DEAD LOAD + LIVE LOAD + WIND LOAD	0.001	-0.000	0.000	0.001	**0.008**	-0.000	0.000
Min rX	115	5 DEAD LOAD + LIVE LOAD + WIND LOAD	0.001	-0.000	-0.000	0.001	**-0.008**	0.000	0.000
Max rY	116	5 DEAD LOAD + LIVE LOAD + WIND LOAD	0.001	-0.000	0.008	0.008	-0.003	**0.002**	-0.000
Min rY	39	5 DEAD LOAD + LIVE LOAD + WIND LOAD	0.001	-0.000	-0.008	0.008	0.003	**-0.002**	-0.000
Max rZ	2	5 DEAD LOAD + LIVE LOAD + WIND LOAD	0.001	-0.003	0.000	0.003	0.003	0.000	**0.001**
Min rZ	20	4 SEISMIC LOAD	0.021	-0.000	-0.000	0.021	-0.000	-0.000	**-0.002**
Max Rst	37	6 DEAD LOAD + LIVE LOAD + SEISMIC LOAD	0.025	-0.002	0.000	**0.025**	0.000	0.000	-0.001

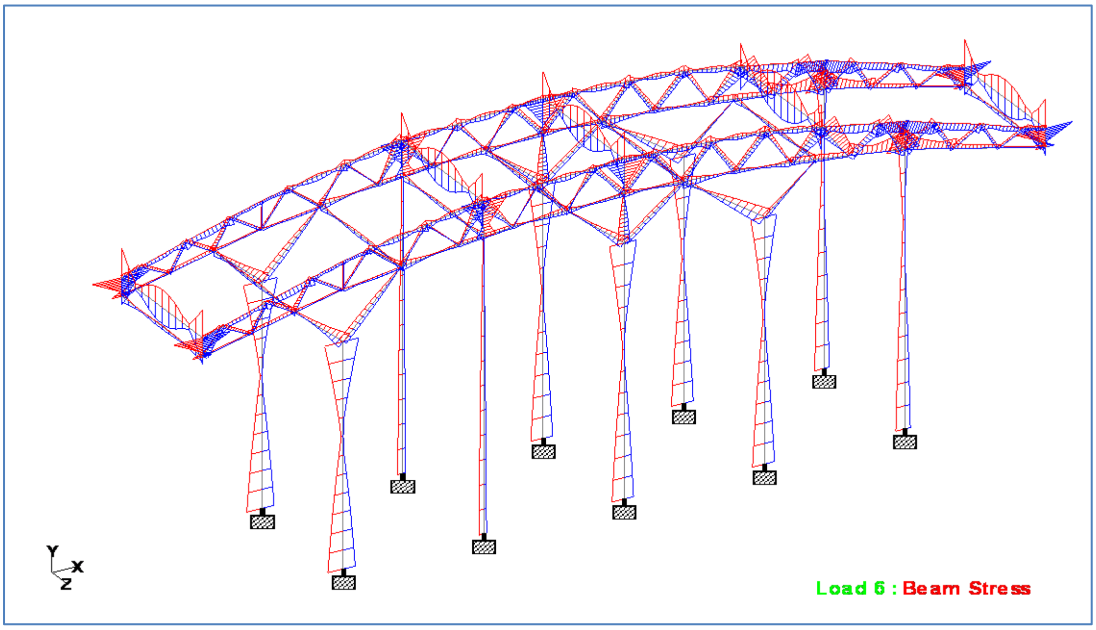

Figure 4.10 Structure beam stresses due to seismic effect

Figure 4.10 indicates the normal stresses distribution on the structural members due to applied seismic effect in the load combination. The maximum stresses acting on the members are shown in table 4.3.

Table 4.3 Summary of maximum beam stresses

Beam	L/C	Length (m)	Max Compressive			Max Tensile		
			Stress kN/m²	Dist. (m)	Corner	Stress kN/m²	Dist. (m)	Corner
1	5	0.937	61735.543	0.937	2	-57835.855	0.937	1
	6	0.937	62121.355	0.937	2	-58348.355	0.937	1
91	5	0.937	62223.074	0.937	2	-58378.270	0.937	1
	6	0.937	62247.004	0.937	2	-58412.355	0.937	1
229	5	0.937	62223.066	0.937	1	-58378.262	0.937	2
	6	0.937	62246.996	0.937	1	-58412.348	0.937	2
282	5	6.000	71177.195	6.000	3	-67070.477	6.000	1
	6	6.000	71177.195	6.000	3	-67070.477	6.000	1
290	**5**	**6.000**	**72103.953**	**0.000**	**3**	**-68002.672**	**0.000**	**1**
	6	**6.000**	**72103.953**	**0.000**	**3**	**-68002.672**	**0.000**	**1**
295	5	6.000	57936.184	0.000	3	-57403.824	0.000	1
	6	6.000	57936.184	0.000	3	-57403.824	0.000	1
296	5	1.794	41249.488	1.794	2	-37565.488	1.794	1
	6	1.794	54112.141	1.794	3	-44063.352	1.794	1
297	5	1.794	41249.488	1.794	1	-37565.488	1.794	2
	6	1.794	54112.141	1.794	4	-44063.352	1.794	2
298	5	6.000	69276.828	0.000	3	-67016.969	0.000	1
	6	6.000	69276.828	0.000	3	-67016.969	0.000	1
299	5	6.000	67184.961	0.000	3	-64845.914	0.000	1
	6	6.000	67184.961	0.000	3	-64845.914	0.000	1

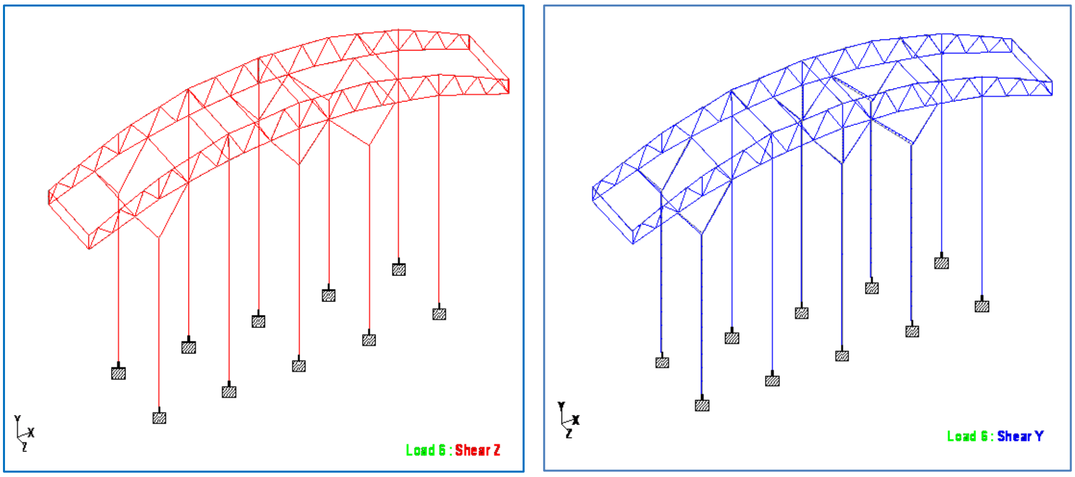

Figure 4.11 Shear force acting along Z and Y axis due to seismic effect

Figure 4.11 indicates the shear forces acting on the structure along the Z and Y axis. For details on maximum shear forces acting on the structural members due to seismic effects on the structure, refer to **(Appendix 4-3)**.

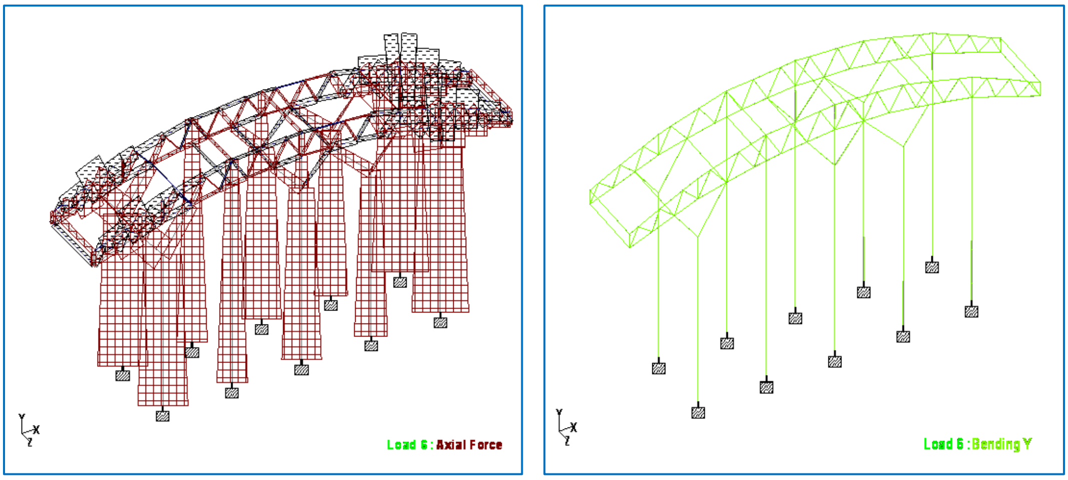

Figure 4.12 Axial force acting along X axis and bending moment along Y axis due to seismic effect

Figure 4.12 indicates the axial force along the X-axis and the bending moment along the Y axis acting on the structure. Refer to **(Appendix 4-3)** for detailed maximum shear forces and bending moment acting on the members.

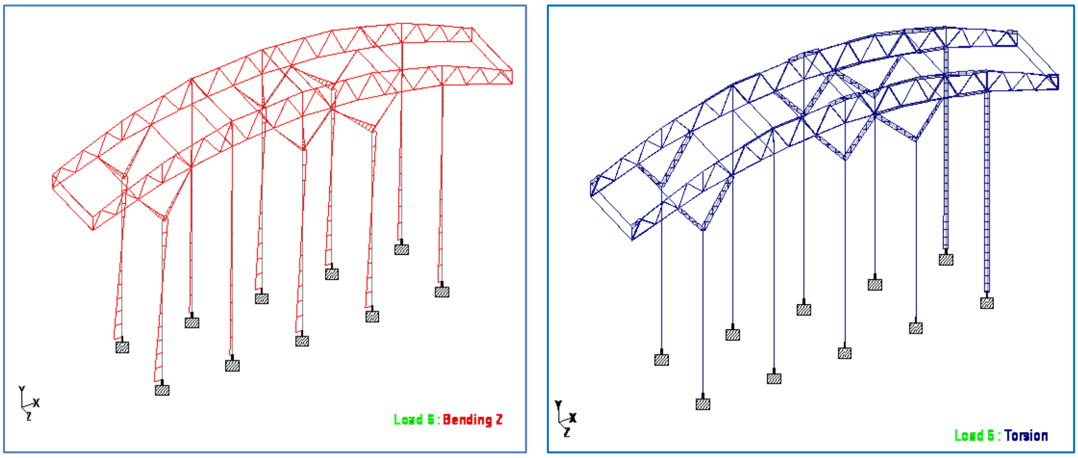

Figure 4.13 Bending moment along Z axis and torsion along X axis due to seismic effect

Figure 4.13 indicates the bending moment along Z-axis and the torsional effects acting on the structure. (**Appendix 4-3)** shows detailed values of the maximum bending moment acting on the members. The seismic loading on the structure caused relatively small torsion as earthquakes travel horizontally in one direction compared to the effect of the wind which swirl.

NONLINERAR DYNAMIC ANAYLSIS RESULTS

Using Time History Analysis Method

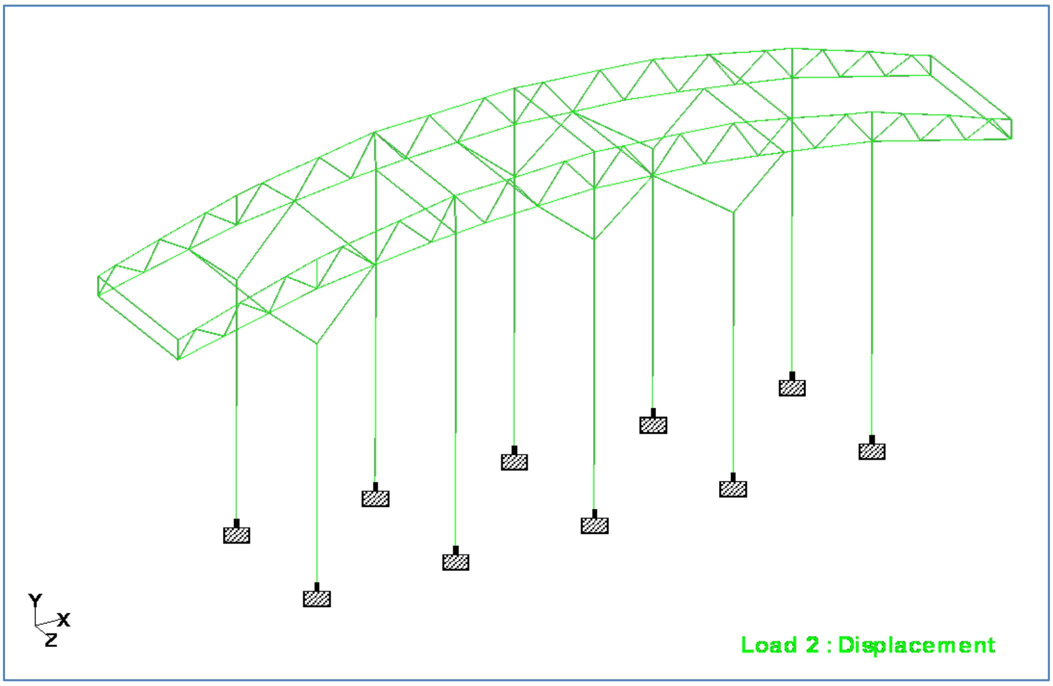

Figure 4.14 Structure nodal displacement due input seismic ground motion

The figure 4.14 indicates the structural nodal displacement due to the induced seismic ground motion effects on the structure. The maximum displacement due to seismic ground motion recorded is 43 mm as shown in **(Appendix 4-5)** or table 4.4

Table 4.4 Summary of maximum nodal displacement

			Horizontal	Vertical	Horizontal	Resultant	Rotational		
	Node	L/C	X m	Y m	Z m	m	rX rad	rY rad	rZ rad
Max X	37	2 Time History Analysis	**0.043**	-0.003	-0.000	0.043	-0.000	0.000	-0.001
Min X	52	1 Static Load	**-0.001**	-0.002	-0.000	0.003	-0.000	-0.000	-0.000
Max Y	2	2 Time History Analysis	0.040	**0.004**	-0.000	0.040	-0.000	0.001	-0.001
Min Y	37	2 Time History Analysis	0.043	**-0.003**	-0.000	0.043	-0.000	0.000	-0.001
Max Z	113	1 Static Load	-0.000	-0.000	**0.004**	0.004	-0.003	-0.001	0.000
Min Z	36	1 Static Load	-0.000	-0.000	**-0.004**	0.004	0.003	0.001	0.000
Max rX	38	1 Static Load	-0.000	-0.000	0.000	0.000	**0.005**	-0.000	0.000
Min rX	115	1 Static Load	-0.000	-0.000	-0.000	0.000	**-0.005**	0.000	0.000
Max rY	52	2 Time History Analysis	0.041	0.001	0.000	0.041	-0.000	**0.001**	0.000
Min rY	129	2 Time History Analysis	0.041	0.001	-0.000	0.041	0.000	**-0.001**	0.000
Max rZ	18	2 Time History Analysis	0.041	-0.000	-0.000	0.041	-0.000	0.000	**0.001**
Min rZ	20	2 Time History Analysis	0.041	-0.000	0.000	0.041	-0.000	0.000	**-0.004**
Max Rs	37	2 Time History Analysis	0.043	-0.003	-0.000	**0.043**	-0.000	0.000	-0.001

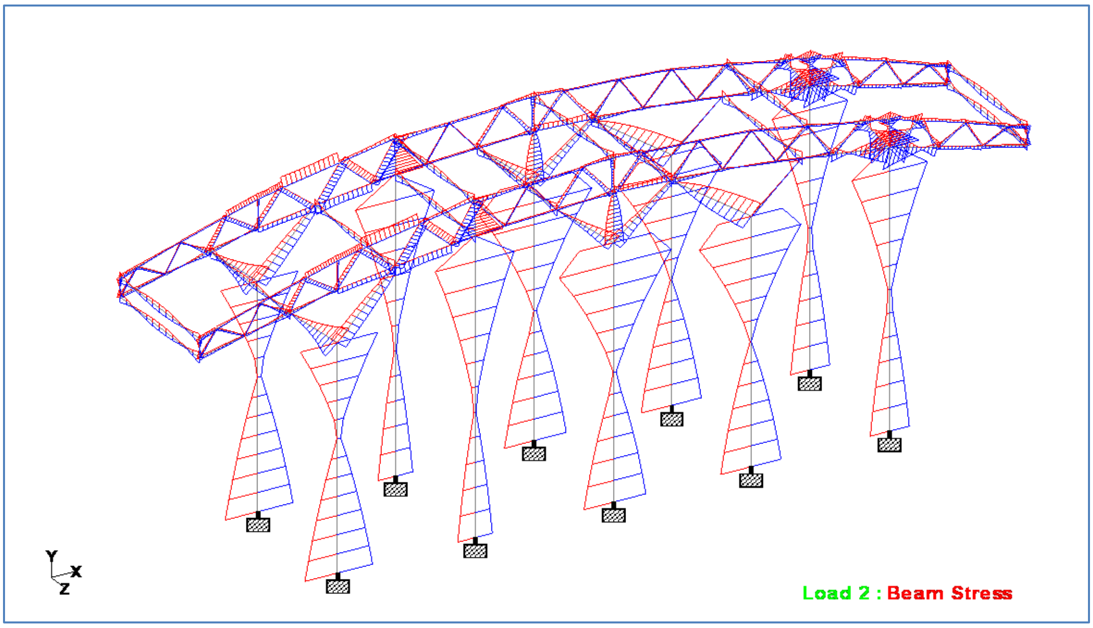

Figure 4.15 Structural beam stresses due input seismic ground motion

Figure 4.15 indicates the normal stresses distribution on the structural members due to the applied seismic ground acceleration effect on the structure. The maximum stresses on the members are shown in table 4.5.

Table 4.5 Summary of maximum beam stresses

Beam	L/C	Length (m)	Max Compressive			Max Tensile		
			Stress kN/m^2	Dist. (m)	Corner	Stress kN/m^2	Dist. (m)	Corner
29	2.Time History analysis	14.717	87483.664	13.491	1	-83840.977	13.491	3
44	2.Time History analysis	12.674	117475.00	11.618	1	-126051.00	11.618	3
60	2.Time History analysis	12.236	112492.00	11.216	1	-108135.00	11.216	3
184	2.Time History analysis	12.674	117475.00	11.618	1	-126051.00	11.618	3
199	2.Time History analysis	12.236	112492.00	11.216	1	-108135.00	11.216	3

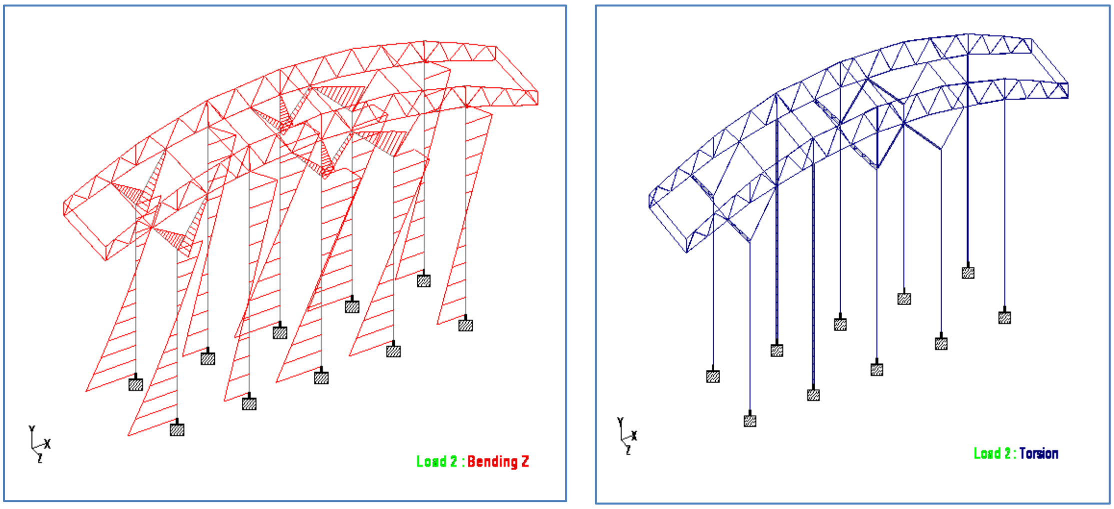

Figure 4.16 Bending moment along Z axis and torsion along X axis due to input seismic ground motion

Figure 4.16 indicates the bending moment along Z-axis and the torsional effects acting on the structure. (**Appendix 4-6)** shows detailed values of the maximum bending moment acting on the members. The seismic ground acceleration caused relatively small torsion about 0.005 rad on the structure.

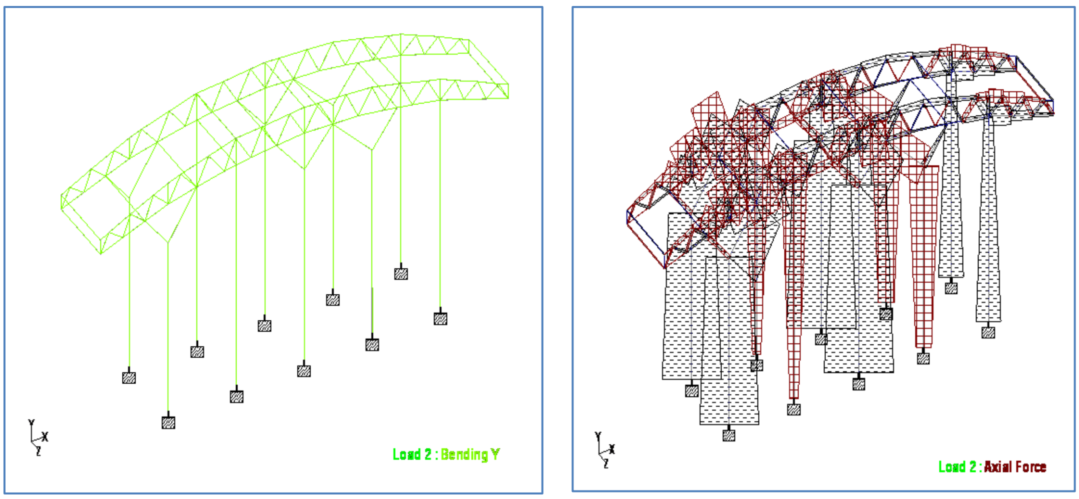

Figure 4.17 Bending moment acting along Y axis and axial force along X-axis due to input seismic ground motion

Figure 4.17 indicates the bending moment along the Y-axis and the axial force along the X-axis acting on the structure. Refer to **(Appendix 4-6)** for detailed maximum shear forces and bending moment acting on the members.

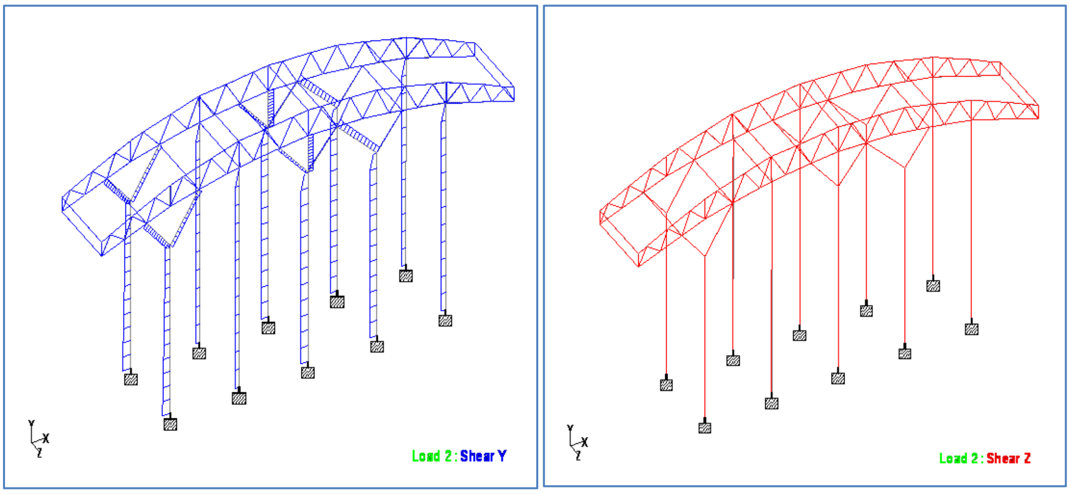

Figure 4.18 Shear force acting along Y and Z axis due to input ground motion

Figure 4.18 indicates the shear forces acting on the structure along the Y-axis and Z-axis. For the maximum shear forces acting on the structure members due to seismic ground acceleration on the structure, refer to **(Appendix 4-6)** for detailed results.

4.10 Discussion and analysis of the results

The Canopy structure designed on STAAD. Pro software is simulated under uniformly distributed loadings in three coordinates system i.e. X, Y and Z. All the basic loads namely dead, live, wind, and seismic have been considered for analysis using both Equivalent Static Analysis (ESA) and Dynamic Analysis (time history analysis) method. Fixed support is assumed with aim to control the allowable deflection (side way).

The 17 m canopy structure is analysed on STAAD Pro using the equivalent static analysis method, the results show that resultant vertical deflection is 25 mm. However, for nonlinear dynamic analysis the resultant nodal displacement is 43 mm. The value of displacement by dynamic analysis method is higher than that of static analysis. The reason for that is practically because dynamic analysis is case sensitive .i.e. it provides more realistic measures of response compare to static analysis method which is more applicable only to regular structure where the structure shall not be unbalanced in its distribution of mass or stiffness. Where there is need to get adequate information on seismic demands imposed by the design ground motion on the structure dynamic analysis is recommended.

Nonlinear dynamic analysis if well implemented, accurate response of the structure will be achieved. It also gives expected performance of the structural systems by estimating its strength and demands in design earthquake. However it should be noted that for effective application of nonlinear dynamic analysis, important consideration must include definition of performance objectives, selection of input ground motions in most cases actual recorded ground motions from past earthquakes and construction of appropriate nonlinear analysis model. With dynamic analysis greater confidence in building performance characteristic including safety would be achieved as it identify nonlinear dynamic response characteristic such as yielding mechanisms associated internal forces and deformation demands. However equivalent static analysis would be preferred for estimation of displacement demands for structures with well balance span and uniformly distributed stiffness where response can be captured by the predominant translational mode of vibrations.

4.10.1 Using Eurocode, ASCE & BS EN 1993 to assess the safety of the Structure

Deflection is a serviceability issue that must be considered along with strength requirements in the design of structures. Deflection limits prescribed in many codes are dependent upon many factors including the type of forces imposing the deflection i.e. wind or seismic, the use of structure, constraints of the structure (adjacent buildings) and finishes attached to or contained inside the structure. Deflection in structures results from both gravity loads and wind or seismic loads that act on buildings. Steel framing are less rigid, they deflect due to loads. Therefore limitations are required to maintain function and appearance of the plaster panels.

In ASTM C926, in Annex A2 on design considerations, it states that maximum allowable deflection for vertical or horizontal for plaster, not including cladding shall be L/360. However, ductile structure such as steel and reinforced concrete may have larger limitations with detriment to strength and performance than brittle building such as unreinforced masonry which requires stringent drift limitation. However, Eurocode **[EN1998. 4.4.3.2(c)]**, stated that for building having non-structural elements fixed in a way that so as not to interfere with structural deformations, or without non-structural elements:

$$d_r \mathrm{v} \leq 0.01\mathrm{h}$$

d_r is the design inter-storey drift, h is the storey height and v is the reduction factor which takes into account the lower return period of the seismic action associated with the damage limitation requirement.(recommended value of v is 0.4 for importance classes III & IV and v is 0.5 for importance classes I & II.

ASCE code states that design story drift (Δ) shall not exceed the allowable story drift (Δ_a). However the code stated that structures with significant torsional deflections, the maximum drift shall include the torsional effects. The allowable deflection drift for seismic group III applied for the structure is 0.01h. For structure to be considered safe, its deflection under applied loadings must not exceed 0.01h. For the British code, the serviceability limits for vertical and horizontal deflections effects in UK

National Annex to **[BS EN 1993-1-1]** gives suggested limits for vertical and horizontal deflections due to variable actions only, stating that deflections due to permanent actions need not verified. With agreement from the client, different limits may be used for a specified project.

Table 4.6 BS EN 1993 vertical deflection limits

Design Situation	Deflection Limits
Cantilevers	$\leq L/180$
Beams carrying plaster of brittle finish	$\leq L/360$
Other beams (except purlins and sheeting rails)	$\leq L/200$

In both Eurocode and BS EN 1993, it is clearly stated that the maximum deflection calculated must not exceed deflection limit. However deflection limits are not given directly in Eurocode 3; instead they stated clearly that reference must be made to the National Annex.

In STAAD Pro, checks for deflection are made independently for each axis. STAAD Pro finds the resultant deflection, d, and compares l/d (length to deflection ratio) against the allowable limit specified by the user. For the analysis, the user defined limit is

$$\text{Safe} < 1 \leq \text{fails} < 1.5 \ \leq \text{Extreme fail (user define condition)}$$

For seismic analysis of a structure, dynamic analysis cannot be performed for any seismic peak ground acceleration of less than 0.1g. However, for this canopy structure the recommended analysis for the structure is static analysis e.g. Equivalent Static Analysis.

Table 4.7 Check for deflection limits

Load Combination (L/C)	Max Deflected values	Deflection Checks		
		BS EN 1993 $\leq \ L/180$	Eurocode $\leq$ 0.01 0 h	ASCE $\leq$ 0.010h_x
Equivalent Static Analysis (Wind effects) (DL + LL + WL)	8 mm	0.008 ≤ 0.094	0.008 ≤ 0.17	0.008 ≤ 0.17
Equivalent Static Analysis (Seismic effect) (DL + LL + EL)	25 mm	0.025 ≤ 0.094	0.025 ≤ 0.17	0.025 ≤ 0.17
Nonlinear Dynamic Analysis Time History analysis	43 mm	0.043 ≤ 0.094	0.043 ≤ 0.17	0.043 ≤ 0.17

For this project, the peak ground acceleration data use in the analysis is 0.000264g (0.00259 cm/sec/sec) for earthquake felt in Malaysia and this has minimal effect on structures. In this case equivalent static analysis results would be considered value. However for example the second Penang Bridge which is 24 km with a lifespan of 120 years with ability to resist an earthquake up to magnitude 7.5 on Richter scale. The bridge is designed for peak ground acceleration for 0.177g and 0.3261g for 2500 years event no collapse under seismic effects. However, ASCE code recommended as given in (table 4.8) the seismic design category based on short period response acceleration.

Table 4.8 ASCE seismic design category based on short period acceleration

Values of S_{DS}	Seismic use groups		
	I	II	III
S_{DS} < 0.167g	A	A	A
0.167 g ≤ S_{DS} < 0.33g	B	B	C
0.33 g ≤ S_{DS} < 0.50g	C	C	D
0.50 g ≤ S_{DS}	D^a	D^a	D^a

In all the analysis performed by the two methods, it is indicated that the structure is safe under the seismic ground motion; however what if the deflection is beyond the limits defined in the provisions, the deflection can be controlled. The simplest way is to increase the geometrical properties/sectional sizes of frames, but it is not advisable as it adds to the tonnage of the whole building, adding not only to the seismic forces but also adding to the cost subsequently. We need a solution wherein the sway of the frame can be controlled and the section sizes are also not increased.

Besides, there are more methods for seismic retrofitting which includes;

- Conventional strengthening methods which includes addition of new structural elements to the systems and enlarging the existing members e.g. concrete jacketing, addition of column members for vertical irregularities
- Traditional methods of seismic retrofitting such as building mass reduction and optimizing structural design
- Retrofit of structure using innovative materials such as use of high performance concrete, high performance steel and fibre reinforced plasters
- Base isolation. This is done by placing flexible isolation systems between the foundation and the superstructure and this would provide safety against collapse

Finally, today due to improving technology, engineers have come up with more innovative approaches for retrofitting such as;

- Stiffness reduction
- Ductility increase
- Damage controlled structure by installation of damping system
- Use of composite materials.

CHAPTER FIVE

CONCLUSION AND RECOMMENDATION

The equivalent static results indicated that the wind effect is minimal on the structure hence the designer has considered wind effects during the canopy design. In Malaysia micro-zonation map, for zone 1 the wind basic velocity is 33.5 m/s^2. The minimum deflection due load combination 5 with wind effects inclusive is 8 mm.

Under the equivalent static analysis, the response of the structure gave a promising result. The canopy's maximum resultant displacement is 25 mm and for nonlinear dynamic analysis, the maximum resultant displacement is 43 mm. The deflection limit check in (BS EN 1993, EUROCODE, and ASCE CODE) indicated that structure is safe under earthquake of the given magnitude.

From the analysis of results above, conclusively the structure is safe under the given earthquake because the structure has been designed to have lateral and vertical force-resisting systems that provide adequate strength, stiffness, and energy dissipation capacity to withstand such earthquake effects felt in the region with the prescribed limits of deformation and strength demand.

The safety of the structure under such an earthquake is attributed to designers' clear understanding of the site soil characteristic, the most occurring seismic peak ground acceleration, and the use of the codes or standards to specify correctly the structural elements. This is always achieved through proper planning, analysis of site parameters and applying good design knowledge. The relevancy of such an objective is, it gives us the guarantee that the safety of the occupants is assured under such unpredictable weather and occurrence of natural disasters.

Finally, it is recommended that despite Malaysia not being located in the active seismic zone, designers must start taking the seismic effects into consideration. Though a legalised Malaysian seismic code has not been documented for seismic effect consideration during design, the PGA can be assumed as 0.2g – 0.25g for special structures e.g. bridge since they are expose to critical conditions. (Adnan et al,.2002) suggested PGA range of 0.1g to 0.167g for major towns in Peninsular Malaysia for 2500 years return period.

References

Giordano A., Guadagnuolo M., Faella G. (2008). *Pushover analysis of plan irregular masonry buildings.14th World conference on Earthquake Engineering, Beijing, China.*

De Stefano M., Pintucchi B., (2002).*A review of research on seismic behaviour of irregular building structures.*

Fujii K., Nakano Y., Sanada Y., (2004). *Simplified nonlinear analysis procedure asymmetric building.13th World Conference on Earthquake Engineering, Vancouver, B.C., Canada.*

ASCE/SEI 7-10 (2010*). Minimum design loads for buildings and other structures, American Society of Civil Engineers.*

Gulkan P. and Sozen M.A. (1974).*Inelastic Response of Reinforced Structure to Earthquake Ground Motion, Journal of the American Concrete Institute, vol 71, p 601-609.*

Rosenblueth .E. and Herrera .I., (1964). *On a Kind of Hysteretic Damping, Journal of Engineering Mechanics Division, ASCE*

Takeda .T.,Sozen M.A. and Nielson N.N., (1970). *Reinforced Concrete Response to Simulated Earthquakes, Journal of Structural Division, ASCE, 96, 2557-2573.*

Chopra A.K., (1995). *Dynamics of Structure Theory and Application to Earthquake Engineering, Prentice Hall, New Jersey.*

Federal Emergency Management Agency, (2002).*Prestandard and Commentary for the Rehabilitation of Buildings, FEMA- 356*

Newmark N.M. and Hall W.J., (1982*). Earthquake Engineering Research Institute, Barkeley, CA.*

Fajfar P. and Fischinger M., (1982).*Nonlinear Seismic Analysis of R/C Buildings: Implications of a case study, European Earthquake Engineering, 31-43.*

Miranda E. and Ruiz G.J.,(2002). *Evaluation of Approximate Methods to Estimate Maximum Inelastic Displacement Demands, Earthquake Engineering and Structural Dynamics, vol 31, 539- 560*

Eurocode 1 & 8, Part 1-4 *Action on Structure, Wind Action and Design of Structures for Earthquake Resistance*

Malaysia Standard (MS 1553:2002)*, Code of practice on wind loading for building structures*

Handriyawan A.and Irsyam M (2002)*, the effect of the lastetst Sumatra earthquake to Malaysian Peninsular*

International Association for Earthquake Engineering (IAEE) manual, *Structural performance during earthquakes*

Chopra A. and Goel K (2004), *a modal pushover analysis procedure to estimate seismic demands for unsymmetrical plan- plan buildings.*

Richard Bradshaw, D. C. (2002, June). *Special Structure: Past Present and Future*.

UTP. (2012, 2). *UTP*. Retrieved from http://www.utp.edu.my/index.php?option=com_content&view=article&id=49&Itemid=1901.

Wikitravel. (2012). *http://wikitravel.org/upload/shared/2/21/KLIA-TERMINAL2.jpg*. Retrieved from KLIA picture.

APPENDICES

Node	X (m)	Y (m)	Z (m)
1	2.144	12.228	0.000
2	2.144	13.165	0.000
3	2.904	13.456	0.000
4	4.134	12.771	0.000
5	4.807	14.139	0.000
6	6.074	13.297	0.000
7	6.814	14.813	0.000
8	8.140	0.000	0.000
9	8.140	11.255	0.000
10	8.143	13.852	0.000
11	8.143	15.245	0.000
12	9.237	15.514	0.000
13	10.630	14.255	0.000
15	11.700	16.015	0.000
17	13.075	14.607	0.000
18	14.087	16.510	0.000
19	14.140	0.000	0.000
20	14.149	14.717	0.000
21	15.401	14.845	0.000
22	16.452	16.610	0.000
23	17.683	14.986	0.000
24	18.856	16.753	0.000
25	20.140	0.000	0.000
26	20.141	15.111	0.000
27	20.141	16.829	0.000
28	20.142	12.674	0.000
29	21.389	16.743	0.000
30	22.658	14.987	0.000
31	23.839	16.611	0.000
32	24.885	14.894	0.000
33	26.135	16.448	0.000
34	26.140	12.236	0.000
35	26.140	0.000	0.000
36	27.218	14.606	0.000
37	28.339	14.442	0.000
38	28.567	15.980	0.000
39	29.676	14.247	0.000
40	31.017	15.452	0.000
41	32.140	0.000	0.000
42	32.141	13.852	0.000
43	32.141	15.245	0.000
44	33.473	14.779	0.000
45	34.224	13.294	0.000
46	35.433	14.135	0.000
47	36.174	12.765	0.000
48	37.317	13.422	0.000
52	38.140	12.228	0.000
52	38.140	12.228	0.000
53	38.140	13.165	0.000
78	2.144	12.228	6.000
79	2.144	13.165	6.000
80	2.904	13.456	6.000
81	4.134	12.771	6.000
82	4.807	14.139	6.000
83	6.074	13.297	6.000
84	6.814	14.813	6.000
85	8.140	0.000	6.000
86	8.140	11.255	6.000
87	8.143	13.852	6.000
88	8.143	15.245	6.000
89	9.237	15.514	6.000
90	10.630	14.255	6.000
92	11.700	16.015	6.000
94	13.075	14.607	6.000
95	14.087	16.510	6.000
96	14.140	0.000	6.000
97	14.149	14.717	6.000
98	15.401	14.845	6.000
99	16.452	16.610	6.000
100	17.683	14.986	6.000
101	18.856	16.753	6.000
102	20.140	0.000	6.000
103	20.141	15.111	6.000
104	20.141	16.829	6.000
105	20.142	12.674	6.000
106	21.389	16.743	6.000
107	22.658	14.987	6.000
108	23.839	16.611	6.000
109	24.885	14.894	6.000
110	26.135	16.448	6.000
111	26.140	12.236	6.000
112	26.140	0.000	6.000
113	27.218	14.606	6.000
114	28.339	14.442	6.000
115	28.567	15.980	6.000
116	29.676	14.247	6.000
117	31.017	15.452	6.000
118	32.140	0.000	6.000
119	32.141	13.852	6.000
120	32.141	15.245	6.000
121	33.473	14.779	6.000
122	34.224	13.294	6.000
123	35.433	14.135	6.000
124	36.174	12.765	6.000
125	37.317	13.422	6.000
129	38.140	12.228	6.000
130	38.140	13.165	6.000

Table 1. Structure nodal coordinates

Node Displacement Summary

	Node	L/C	X (m)	Y (m)	Z (m)	Resultant (m)	rX (rad)	rY (rad)	rZ (rad)
Max X	37	6:DEAD LOAD	**0.025**	-0.002	0.000	0.025	0.000	0.000	-0.001
Min X	52	1:DEAD LOAD	**-0.001**	-0.003	-0.000	0.003	-0.000	-0.000	-0.001
Max Y	1	4:SEISMIC LO.	0.023	**0.001**	-0.000	0.023	-0.000	-0.000	-0.000
Min Y	130	6:DEAD LOAD	0.022	**-0.005**	-0.000	0.023	-0.003	0.001	-0.001
Max Z	116	5:DEAD LOAD	0.001	-0.000	**0.008**	0.008	-0.003	0.002	-0.000
Min Z	39	5:DEAD LOAD	0.001	-0.000	**-0.008**	0.008	0.003	-0.002	-0.000
Max rX	38	5:DEAD LOAD	0.001	-0.000	0.000	0.001	**0.008**	-0.000	0.000
Min rX	115	5:DEAD LOAD	0.001	-0.000	-0.000	0.001	**-0.008**	0.000	0.000
Max rY	116	5:DEAD LOAD	0.001	-0.000	0.008	0.008	-0.003	**0.002**	-0.000
Min rY	39	5:DEAD LOAD	0.001	-0.000	-0.008	0.008	0.003	**-0.002**	-0.000
Max rZ	2	5:DEAD LOAD	0.001	-0.003	0.000	0.003	0.003	0.000	**0.001**
Min rZ	20	4:SEISMIC LO.	0.021	-0.000	-0.000	0.021	-0.000	-0.000	**-0.002**
Max Rst	37	6:DEAD LOAD	0.025	-0.002	0.000	**0.025**	0.000	0.000	-0.001

Table 2. Nodal displacement summary for equivalent static analysis

Beam Displacement Detail Summary

Displacements shown in italic indicate the presence of an offset

	Beam	L/C	d (m)	X (m)	Y (m)	Z (m)	Resultant (m)
Max X	288	6:DEAD LOAD	3.000	**0.025**	-0.004	0.000	0.026
Min X	289	5:DEAD LOAD	3.000	**-0.002**	-0.006	-0.000	0.007
Max Y	1	4:SEISMIC LO.	0.094	0.023	**0.001**	0.000	0.023
Min Y	295	5:DEAD LOAD	3.000	0.000	**-0.025**	-0.000	0.025
Max Z	205	5:DEAD LOAD	1.242	0.001	-0.000	**0.008**	0.008
Min Z	67	5:DEAD LOAD	1.242	0.001	-0.000	**-0.008**	0.008
Max Rst	295	6:DEAD LOAD	3.000	0.023	-0.025	-0.000	**0.034**

Table 3. Beam displacement details summary for equivalent static analysis

Beam End Displacement Summary

Displacements shown in italic indicate the presence of an offset

	Beam	Node	L/C	X (m)	Y (m)	Z (m)	Resultant (m)
Max X	64	37	6:DEAD LOAD	**0.025**	-0.002	0.000	0.025
Min X	87	52	1:DEAD LOAD	**-0.001**	-0.003	-0.000	0.003
Max Y	1	1	4:SEISMIC LO.	0.023	**0.001**	0.000	0.023
Min Y	*89*	*53*	*6:DEAD LOAD*	*0.022*	***-0.005***	*0.000*	*0.023*
Max Z	205	116	5:DEAD LOAD	0.001	-0.000	**0.008**	0.008
Min Z	67	39	5:DEAD LOAD	0.001	-0.000	**-0.008**	0.008
Max Rst	64	37	6:DEAD LOAD	0.025	-0.002	0.000	**0.025**

Table 4. Beam end displacement summary for equivalent static analysis

Beam End Force Summary

The signs of the forces at end B of each beam have been reversed. For example: this means that the Min Fx entry gives the la value for an beam.

				Axial	Shear		Torsion	Bending	
	Beam	Node	L/C	Fx (kN)	Fy (kN)	Fz (kN)	Mx (kNm)	My (kNm)	Mz (kNm)
Max Fx	16	8	5:DEAD LOAD	**109.752**	0.825	0.354	-0.093	-1.834	4.858
Min Fx	78	43	6:DEAD LOAD	**-108.052**	-2.559	0.064	0.219	-0.301	1.111
Max Fy	56	34	6:DEAD LOAD	23.279	**17.037**	-0.031	1.321	-0.283	47.503
Min Fy	133	9	6:DEAD LOAD	57.835	**-10.825**	-0.451	1.063	-0.084	24.983
Max Fz	1	1	5:DEAD LOAD	9.577	0.797	**9.998**	-0.113	-2.401	0.111
Min Fz	151	78	5:DEAD LOAD	9.577	0.797	**-9.998**	0.113	2.401	0.111
Max Mx	75	41	5:DEAD LOAD	104.521	-0.135	0.645	**1.658**	-6.047	-0.405
Min Mx	213	118	5:DEAD LOAD	104.521	-0.135	-0.645	**-1.658**	6.047	-0.405
Max My	185	105	5:DEAD LOAD	35.133	-0.462	-3.474	-0.046	**9.217**	-1.060
Min My	45	28	5:DEAD LOAD	35.133	-0.462	3.474	0.046	**-9.217**	-1.060
Max Mz	16	8	6:DEAD LOAD	94.141	10.709	0.354	-0.093	-1.834	**71.312**
Min Mz	16	9	6:DEAD LOAD	69.939	10.709	0.354	-0.093	2.145	**-49.223**

Table 5. Beam end force summary for equivalent static analysis

Beam Force Detail Summary

Sign convention as diagrams:- positive above line, negative below line except Fx where positive is compression. Distance d is giv beam end A.

				Axial	Shear		Torsion	Bending	
	Beam	L/C	d (m)	Fx (kN)	Fy (kN)	Fz (kN)	Mx (kNm)	My (kNm)	Mz (kNm)
Max Fx	16	5:DEAD LOAD	0.000	**109.752**	0.825	0.354	-0.093	-1.834	4.858
Min Fx	78	6:DEAD LOAD	1.411	**-108.052**	-2.559	0.064	0.219	-0.301	1.111
Max Fy	56	6:DEAD LOAD	0.000	23.279	**17.037**	-0.031	1.321	-0.283	47.503
Min Fy	133	6:DEAD LOAD	2.905	57.835	**-10.825**	-0.451	1.063	-0.084	24.983
Max Fz	1	5:DEAD LOAD	0.000	9.577	0.797	**9.998**	-0.113	-2.401	0.111
Min Fz	151	5:DEAD LOAD	0.000	9.577	0.797	**-9.998**	0.113	2.401	0.111
Max Mx	75	5:DEAD LOAD	0.000	104.521	-0.135	0.645	**1.658**	-6.047	-0.405
Min Mx	213	5:DEAD LOAD	0.000	104.521	-0.135	-0.645	**-1.658**	6.047	-0.405
Max My	185	5:DEAD LOAD	0.000	35.133	-0.462	-3.474	-0.046	**9.217**	-1.060
Min My	45	5:DEAD LOAD	0.000	35.133	-0.462	3.474	0.046	**-9.217**	-1.060
Max Mz	16	6:DEAD LOAD	0.000	94.141	10.709	0.354	-0.093	-1.834	**71.312**
Min Mz	16	6:DEAD LOAD	11.255	69.939	10.709	0.354	-0.093	2.145	**-49.223**

Table 6. Beam force details summary for equivalent static analysis

Node Displacement Summary

	Node	L/C	X (m)	Y (m)	Z (m)	Resultant (m)	rX (rad)	rY (rad)	rZ (rad)
Max X	37	2:Time History	**0.043**	-0.003	-0.000	0.043	-0.000	0.000	-0.001
Min X	52	1:Static Load	**-0.001**	-0.002	-0.000	0.003	-0.000	-0.000	-0.000
Max Y	2	2:Time History	0.040	**0.004**	-0.000	0.040	-0.000	0.001	-0.001
Min Y	37	2:Time History	0.043	**-0.003**	-0.000	0.043	-0.000	0.000	-0.001
Max Z	113	1:Static Load	-0.000	-0.000	**0.004**	0.004	-0.003	-0.001	0.000
Min Z	36	1:Static Load	-0.000	-0.000	**-0.004**	0.004	0.003	0.001	0.000
Max rX	38	1:Static Load	-0.000	-0.000	0.000	0.000	**0.005**	-0.000	0.000
Min rX	115	1:Static Load	-0.000	-0.000	-0.000	0.000	**-0.005**	0.000	0.000
Max rY	52	2:Time History	0.041	0.001	0.000	0.041	-0.000	**0.001**	0.000
Min rY	129	2:Time History	0.041	0.001	-0.000	0.041	0.000	**-0.001**	0.000
Max rZ	18	2:Time History	0.041	-0.000	-0.000	0.041	-0.000	0.000	**0.001**
Min rZ	20	2:Time History	0.041	-0.000	0.000	0.041	-0.000	0.000	**-0.004**
Max Rst	37	2:Time History	0.043	-0.003	-0.000	**0.043**	-0.000	0.000	-0.001

Table 7. Beam displacement summary for time history analysis

Beam Displacement Detail Summary

Displacements shown in italic indicate the presence of an offset

	Beam	L/C	d (m)	X (m)	Y (m)	Z (m)	Resultant (m)
Max X	44	2:Time History	11.407	**0.044**	0.000	0.000	0.044
Min X	289	1:Static Load	3.000	**-0.001**	-0.002	0.000	0.002
Max Y	282	2:Time History	3.000	0.043	**0.005**	-0.000	0.043
Min Y	295	1:Static Load	3.000	-0.001	**-0.015**	-0.000	0.015
Max Z	205	1:Static Load	1.490	-0.000	-0.000	**0.005**	0.005
Min Z	67	1:Static Load	1.490	-0.000	-0.000	**-0.005**	0.005
Max Rst	289	2:Time History	3.000	0.044	0.002	0.000	**0.044**

Table 8. Beam displacement details summary for time history analysis

Beam End Displacement Summary

Displacements shown in italic indicate the presence of an offset

	Beam	Node	L/C	X (m)	Y (m)	Z (m)	Resultant (m)
Max X	64	37	2:Time History	**0.043**	-0.003	-0.000	0.043
Min X	87	52	1:Static Load	**-0.001**	-0.002	-0.000	0.003
Max Y	1	2	2:Time History	0.040	**0.004**	-0.000	0.040
Min Y	64	37	2:Time History	0.043	**-0.003**	-0.000	0.043
Max Z	198	113	1:Static Load	-0.000	-0.000	**0.004**	0.004
Min Z	59	36	1:Static Load	-0.000	-0.000	**-0.004**	0.004
Max Rst	64	37	2:Time History	0.043	-0.003	-0.000	**0.043**

Table 9. Beam end displacement details summary for time history analysis

Beam End Force Summary

The signs of the forces at end B of each beam have been reversed. For example: this means that the Min Fx entry gives the la value for an beam.

	Beam	Node	L/C	Axial Fx (kN)	Shear Fy (kN)	Shear Fz (kN)	Torsion Mx (kNm)	Bending My (kNm)	Bending Mz (kNm)
Max Fx	23	15	2:Time History	**116.888**	-0.269	-0.010	-0.005	-0.011	-0.244
Min Fx	44	25	2:Time History	**-139.182**	61.190	-0.159	-0.036	1.022	379.429
Max Fy	16	8	2:Time History	-118.509	**69.554**	-0.184	0.014	1.036	415.373
Min Fy	45	26	2:Time History	-44.661	**-49.960**	-0.510	-0.201	0.230	1.942
Max Fz	91	52	1:Static Load	5.606	-0.148	**5.603**	0.105	-0.988	-0.066
Min Fz	229	129	1:Static Load	5.606	-0.148	**-5.603**	-0.105	0.988	-0.066
Max Mx	75	41	1:Static Load	35.468	-0.248	0.590	**1.464**	-6.672	-2.033
Min Mx	213	118	1:Static Load	35.468	-0.248	-0.590	**-1.464**	6.672	-2.033
Max My	185	105	1:Static Load	13.232	0.105	-2.960	-0.040	**8.394**	0.256
Min My	45	26	1:Static Load	13.232	0.105	2.960	0.040	**-8.394**	0.256
Max Mz	16	8	2:Time History	-118.509	69.554	-0.184	0.014	1.036	**415.373**
Min Mz	16	9	2:Time History	-92.433	43.478	-0.184	0.014	-1.035	**-210.533**

Table 10. Beam displacement details summary for time history analysis

Beam Force Detail Summary

Sign convention as diagrams:- positive above line, negative below line except Fx where positive is compression. Distance d is gi beam end A.

	Beam	L/C	d (m)	Axial Fx (kN)	Shear Fy (kN)	Shear Fz (kN)	Torsion Mx (kNm)	Bending My (kNm)	Bending Mz (kNm)
Max Fx	23	2:Time History	0.000	**116.888**	-0.269	-0.010	-0.005	-0.011	-0.244
Min Fx	44	2:Time History	0.000	**-139.182**	61.190	-0.159	-0.036	1.022	379.429
Max Fy	16	2:Time History	0.000	-118.509	**69.554**	-0.184	0.014	1.036	415.373
Min Fy	45	2:Time History	2.438	-44.661	**-49.960**	-0.510	-0.201	0.230	1.942
Max Fz	91	1:Static Load	0.000	5.606	-0.148	**5.603**	0.105	-0.988	-0.066
Min Fz	229	1:Static Load	0.000	5.606	-0.148	**-5.603**	-0.105	0.988	-0.066
Max Mx	75	1:Static Load	0.000	35.468	-0.248	0.590	**1.464**	-6.672	-2.033
Min Mx	213	1:Static Load	0.000	35.468	-0.248	-0.590	**-1.464**	6.672	-2.033
Max My	185	1:Static Load	0.000	13.232	0.105	-2.960	-0.040	**8.394**	0.256
Min My	45	1:Static Load	0.000	13.232	0.105	2.960	0.040	**-8.394**	0.256
Max Mz	16	2:Time History	0.000	-118.509	69.554	-0.184	0.014	1.036	**415.373**
Min Mz	44	2:Time History	11.407	-112.750	61.194	-0.159	-0.036	-0.790	**-318.561**

Table 11. Beam force details summary for time history analysis