FINITE ELEMENT ANALYSIS OF A GEODESIC DOME STRUCTURE BY USING STAADPRO SOFTWARE

MOHAMAD AIDIL BIN ABD KASSIM
(AA12033)

Thesis submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Civil Engineering

Faculty of Civil Engineering and Earth Resources
Universiti Malaysia Pahang

JANUARY 2015

ABSTRACT

Finite element analysis (FEA) has become common place in recent years, and is now widely used in the construction industry during analysis and design of a structure. Numerical solutions to very complicated stress problems can now be obtained routinely using FEA. Dome structure is unique due to the ability to withstand a larger span with minimum interference. Geodesic domes are efficient structure because it consists of triangular element along the hemisphere to resist the load. Triangle element is the most effective shape in truss and the combination between triangle element and dome make geodesic dome a strong structure to resist load. STAAD.Pro software is based on the finite element analysis and this software widely used in industry because it accuracy during analysis a structure by using finite element method (FEM). This study is about analysis the structural behaviour of the Geodesic Dome Structure by using STAAD.Pro Software to determine the maximum deflection and all the reaction force on the structure when the load is applied to the model.

ABSTRAK

Analisis unsur terhingga (FEA) telah kerap digunakan pada tahun-tahun kebelakangan ini, dan ia kini digunakan secara meluas dalam industri pembinaan dalam analisis dan rekabentuk struktur. Penyelesaian berangka untuk masalah tekanan yang sangat rumit kini boleh didapati secara menggunakan FEA. Struktur kubah adalah unik kerana keupayaan untuk menahan span yang lebih besar dengan gangguan minimum. Kubah geodesi adalah struktur berkesan kerana ia terdiri daripada unsur segi tiga bersama-sama hemisfera untuk menahan beban. Unsur segi tiga adalah bentuk yang paling berkesan dalam kekuda dan kombinasi antara unsur segi tiga dan kubah kubah geodesi menjadi struktur yang kuat untuk menahan beban. Perisian STAAD.Pro adalah berdasarkan kepada analisis unsur terhingga dan perisian ini digunakan secara meluas dalam industri kerana ketepatannya dalam analisis struktur dengan menggunakan kaedah unsur terhingga (FEM). Kajian ini adalah mengenai analisis kelakuan struktur Struktur Geodesic Dome dengan menggunakan Perisian STAAD.Pro untuk menentukan lenturan maksimum dan semua reaksi tekanan pada struktur apabila beban dikenakan kepada model.

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURE

CHAPTER 1

INTRODUCTION

`1.1 BACKGROUND OF STUDY

A dome is a rounded vault made or curved segment or a shell or revolution known as arch rotated around its central vertical exis. The word 'dome' is come from the greek and latin domus which was used up through the renaissace to label a revered house due to the shape of its roof. Dome are stable during construction as each level is made a complete and self supporting ring. Domes achieve heir shape by extending each horizontal layer inward slightly farther than the previous one until they meet at the tope.

A dome is a structural system that consist of one or more layers of element that are arched in all direction and rotated around it central vertical axis. The surface of a dome may be a part of a single surface such as a spehere or a paraboloid, and it may consist of a patchwork or different surfaces. There are many type of dome structure such as ;

a)Ribbed Dome

This is a ribbed dome that consist of a number of intersecting ribs and ring. A rib is a group of element that lie among a meridional line and a ring is a group of elements that constitute a horizontal polygon. A ribbed dome will not be structurally stable unless it is designed as a rigidly jointed system. When the number of ribs is large, it would lead to a problem at the dome structure. There a many type of dome that have been modified from a ribbed dome to another type of dome to improve the load resisting action.

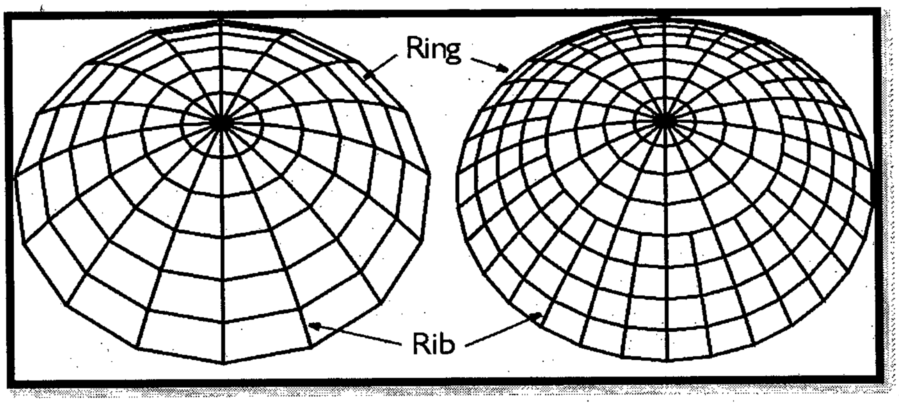

Figure 1.1: Ribbed Dome (Source: www.Geodesic-Unlimited.com)

b)Lamella Dome

Another type of dome are Lamella Dome. This type of dome has diagonals extending from the crown down towards the equator of the dome. The diagonals that extending are in both direction which is clockwise and anti clockwise direction. Some of Lamella dome has horizontal ring and other has not horizontal ring and also has no meridional ribs. The example of lamella dome are as shown in figure below.

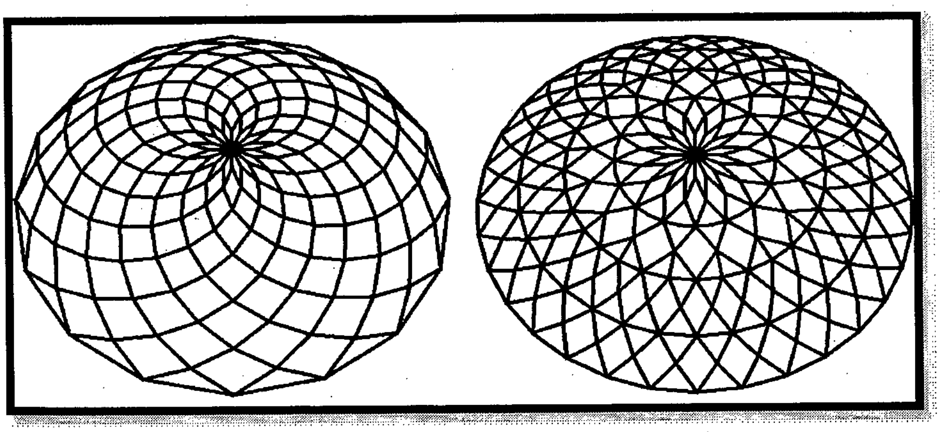

Figure1.2: Lamella Dome (Source: www.Geodesic-Unlimited.com)

c)Schwedler Dome

Schwedler dome has ribs that extending down from the crown of the dome and its rings extending horizontally around the dome. The diagonals of schwedler dome are extending from intersection between ribs and rings on one horizontal ring to those on the next. There are two type of schwedler dome which is normal schwedler dome and the other one is trimmed schwedler dome that are modified from the normal one.

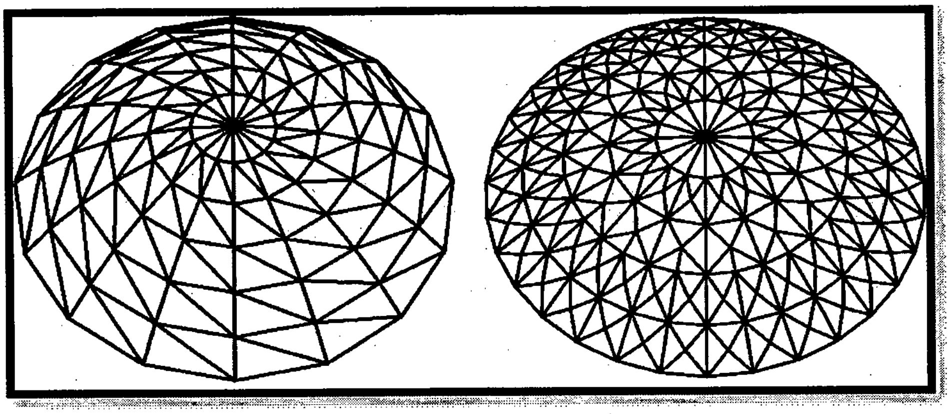

Figure 1.3: Schwedler Dome (Source: www.Geodesic-Unlimited.com)

d)Diamatic Dome

Diamatic Dome is a dome that can be describe as pie-shaped sector repeated radially around the crown of a dome. In diamatic dome, the apex of each sector has a width of zero and at its base, a sectir is 360 degrees divided by the number of sectors that exist.There are two type of dome which is diamatic dome and modified diamatic dome.

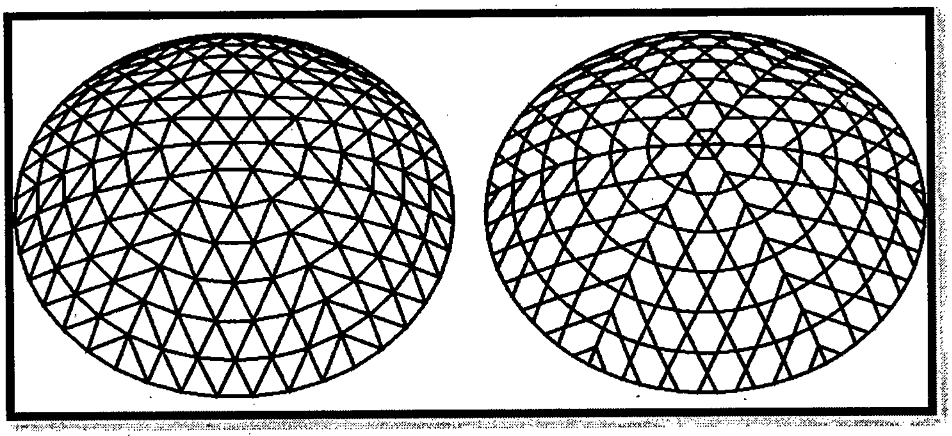

Figure 1.4: Diamatic Dome (Source: www.Geodesic-Unlimited.com)

e) Geodesic Dome

This dome is rather different in its origin, It is derived from one of the platonic or Archimedean solids. Geodesic dome also derived from a prism or anti-prism.A geodesic dome is a spherical or partial-spherical shell structure or lattice shell based on a network of great circles on the surface of a sphere. The geodesic intersect to form triangular rigidity and also distribute the stress acroos the structure. Geodesic dome design begins with icosahedron in a sphere combined with each other forming a hemisphere.

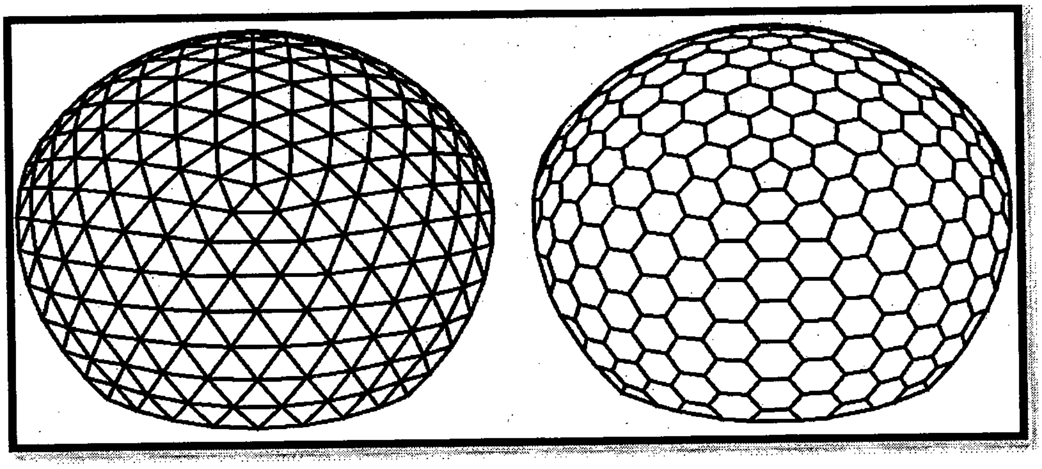

Figure 1.5: Geodesic Dome (Source: www.Geodesic-Unlimited.com)

However, the structureal behaviour of a dome are complex to be understand. The effect of the dome structure and the behaviour of the truss supporting structure are hard to be predicted when load are applied in a direction. This is because the structure are complex allign structure and the positioning factor impact during predicting the deflection, displacement, compression and tension member during a certain load apply at a point in a dome. The structural integrity between the member in geodesic dome structure lead to the stability and variability of reaction acting on every member.

Among the various numerical methods, finite element is very popular and widely used for solving engineering problem. STAAD Pro is a general purpose finite element modeling for numerically solving wide variety of structure analysis problems which include static structural analysis for both linear and non-linear analysis. In general, a finite element solution of structure using Staad Pro may be broken into two stages which are modelling mode and post processing mode. In modeling mode, model will build to define key points, element type and material as required. In this mode also, the loads and constraints(support) will specify and finally solve the resulting set of equations by using specific standard code of practice. In post processing mode, further processing and viewing of the results will see such as the lists of nodal displacements, element forces and moments and deflection plots.

In this study, a structure of a geodesic dome with 20 metre in diameter and 10 metre height is used to analyze the deflection, displacement and stress of every element in the steel frame structure. Finite element method, (FEM) is used in this study in order to get a more accurate result for the behaviour of the dome structure. The dome is model out by using Rhinoceros 4 in order to analyze the behaviour of the dome under various conditions. Various loading vertical and horizontal load is applied to the dome structure. Contour diagram of deflection and stress distribution and result of the analysis is clearly show and stated in STAAD Pro result.

1.2 PROBLEM STATEMENT

Now day, dome structure is widely used by architect due to the artistic value and it is famous because it can produce a large hall or assembly area and reduce the amount of column in between the space. Finite element method is very widely used method for solving engineering problem or to analyze dome structure. In the real situation without calculation by solfware, it might be difficulties to the engineer to calculate all the calculation start from the foundation until the roof. The calculation are consists of the long calculation in order to make sure the design is safe and economical. Manual calculation can cause a lot of mistakes. In this case, STAAD Pro software is used to analyse geodesic dome structure with 20 meter in diameter.

The introduction of STAAD Pro software has resulted in considerable advances in the analysis and design of any structures. Therefore, it is of great encouragement to study and understand the use of software in solving the practical problem. STAAD Pro is use to define the strength behaviour of the element in the dome structure by modeling and analyze the dome structure.

1.3 OBJECTIVES

The objective in analysis the geodesic dome structure by using the STAAD Pro software are:

i. To study the factor that will affect the behaviour of dome structure through a series of analysis. Effect of wind load, selfweight of steel structure will consider as the load acting to the dome truss.
ii. To determine the force and moment value on geodesic dome element when the load is applied.
iii. To determine the deflection and stress value on geodesic dome element when the load is applied using finite element method by STAAD Pro software and design the geodesic dome according tu Eurocode 3, Design of Steel Structure (EN1993-1-1:2005)

1.4 SCOPE OF STUDY

In order to achieve the objective of the study, learning how to use the STAAD Pro program through internet is start. A few tutorials in website Bently Community Forum is doing to practice. Practicing of tutorials can help to solve problem when running the models. Model of dome structure are designed by Rhinocheros 4 software. Finite element analysis is used to calculate the all the reaction parameters which the parameters must success to get result.

This study will focuses on the effect of the various load that acting on the geodesic dome structure using STAAD Pro. A series of analysis with different load applied will run to get the deflection, displacement and stress distribution. A report of result will obtained to know the parameter. The result obtain from the analysis will be used to conclude the behaviour of the dome structure against the load.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Dome is structural system that can withstand a larger span with minimum interference between supports at the end of this structure. Geometry of a dome support itself without needing internal columns or interior load bearing walls. Geodesic dome was idealized and invented by R. Buckminster Fuller in year of 1954. The first geodesic domes that have been designed was built in 1922 by Walter Bauersfeld in Jena, Germany (T. Davis, 2004). Geodesic domes are efficient structures in several ways because the triangle is a very stable shape in resisting force. When a load is applied to the corner of a rectangle, it can deform into a parallelogram but the same force will not deform a triangle. This make a geodesic dome building able to resist force such as snow, earthquake, wind and tornado.

Dome structure are usually used as the top of mosque and church because it has high aesthetic value while geodesic dome are usually used as greenhouse structure, storage and arenas such as tacoma dome in washington, United State Of America. Engineer and architect have the interest in a structure that can cover a large span with minimum interference (M.Kubik, 2009). In this case, a geodesic dome have a pattern that very unique in term of structure arrangement making the geodesic dome are already high aesthetical value before it decorated by any plate that use as the shell of the dome.

2.2 SHAPE

The original shape of geodesic dome was constructed using dodecahderon and octahedron symmetry system. Both symmetry systems was part of the platonic solid that formerly known as polyhydra. Polyhydra are come from a shape of identical polygon. If icosahedron shapes such as shown in figure 2.1 below explode on a sphere, it will result on a sphere that contains 23 equilateral spherical triangles. The vertex that formed can also be describes by the intersection of three great circle where the circles having the same diameter and equal to the sphere. If the lines that join all the vertices are straight lines and formed the shape of triangle around a hemisphere such as shown in figure 2.2, it will create a structure of a geodesic dome structure (M.Kubik 2009).

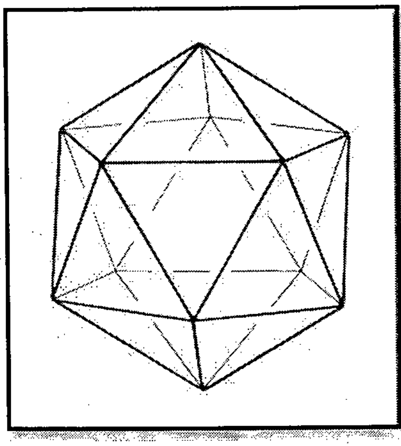

Figure 2.1 Icosahedron

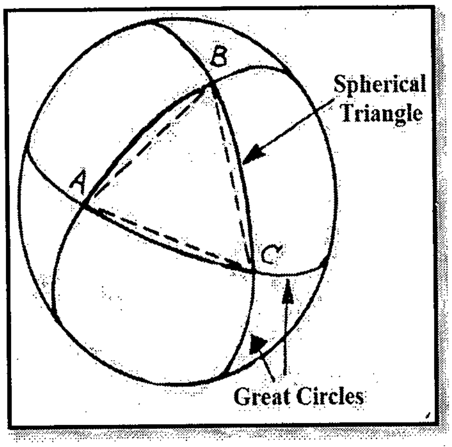

Figure 2.2 **Great Circles**

Source : M.Kubik 2009

In geodesic dome structure, the arrangements of element inside the triangular element are divided into two which is Class I sub-division and Class 2 sub-division. In

Class I sub-division, the line that dividing the triangle are parallel to the edges of primary bracing while in Class II sub-division, the line that dividing the triangle are perpendicular to the primary bracing (Motro, 1984). The arrangement of element that dividing the triangle of geodesic dome are shown on figure 2.3 and figure 2.4 below for Class I and Class II sub-division.

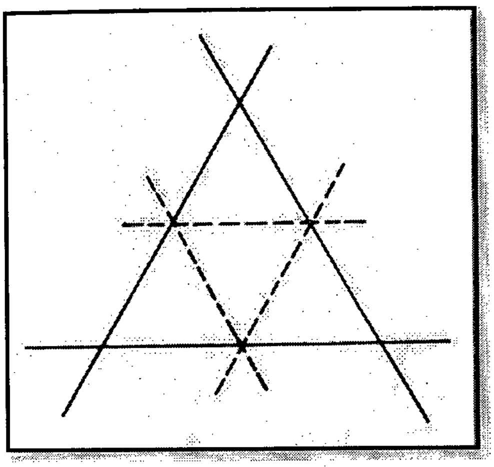

Figure 2.3 Class I Sub- Division

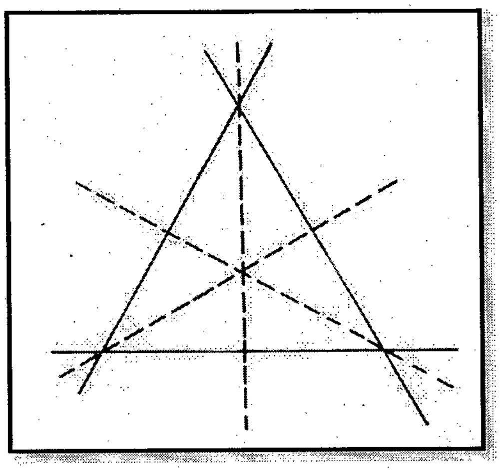

Figure 2.4 Class II Sub Division

Source: M.Kubik 2009

Individual element length for sub-division Class II is greater than the individual element in Class I sub-division. However, the overall member length on a primary bracing for sub-division I is greater than subdivision II. The frequency of the geodesic dome structure is defined by the number of triangle in each edge of primary bracing of geodesic dome.

The frequency of a geodesic dome is usually labelled with alphabet 'V' after the frequency of the element (M. Kubik, 2009). In geodesic dome, only even order frequency can produce a hemispherical shape. The figure 2.5 and figure 2.6 below show an example of the type of geodesic domes and its label according to the frequency

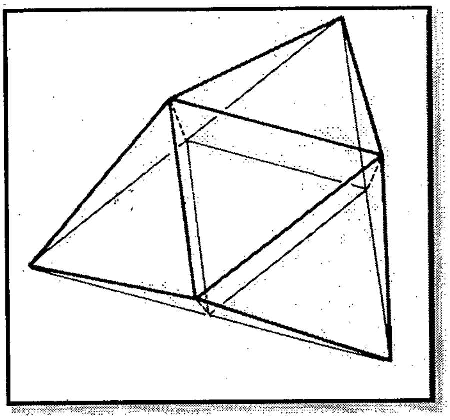

Figure 2.5 2V Dome

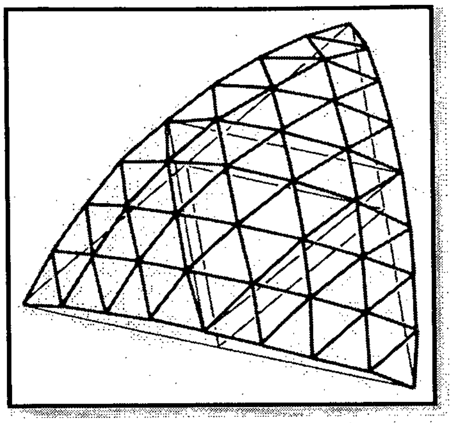

Figure 2.6 8V Dome

Source: M.Kubik 2009

2.3 STIFFNESS METHOD

The stiffness method allows us to analyse a complex structure by breaking the structural member component into element and nodes. Element in this method represent the member while the nodes represent the join between the members along the structure. This method can be used to design a space frame structure and other complex arrangement structure such as geodesic dome. The stiffness method for analyse structure limited to pin jointed element. Every member has related properties to spring, an axial load 'F' can be applied to the end of spring and internally carried resulting in the member length change 'u'. The Relationship between applied load and the resulting displacement is a property of element known as stiffness 'k' and it can be expressed as ;

$$F = ku \tag{1}$$

In order to find the structure response, determine the forces and displacement, a solution to find the stiffness of each member must be found (D.L. Logan, 2012). There are many methods to solve the unknown from a series of simultaneous equation. In

STAAD.Pro, the element stiffness matrices are assembled into a global stiffness matrix by standard matrix techniques used in Finite Element Analysis (FEA) programs. The Technique used by staad pro was similarly from SAP IV where the stiffness matrix is write as:

$$[K] = [LT][D][L] \tag{2}$$

Which is a modified Gauss method

$$[K]\{d\} = \{f\} \tag{3}$$

Becomes

$$[LT][D][L]\{d\} = \{f\} \tag{4}$$

The equation above can be manipulated into a forward and a backward substitution step to obtain$\{d\}$. STAAD can detect singular matrices and solve the matrices through basic solver and Advance Solver.

In basic solver, the method of decomposition is used as an approach that is particular and suited for the structural analysis. Since the stiffness matrices of all linearly elastic structure are always symmetric, Modified Cholesky's method is applied in this problem.

This method is reasonable accurate and suited for the Gaussian elimination process in solving the simultaneous equations. Another approach is by using "Advance Solver" technique that is mathematically equivalent to the modified Cholesky's method. However, the order of operation, memory use and file use is highly optimized and the run times are usually 10 to 100 times faster.

2.4 FINITE ELEMENT METHOD (FEM)

Finite element is a numerical method to solve the engineering problems. It is a very powerful method in performing linear and non-linear problem in structural analysis. Finite Element analysis was first developed in 1943 by R.Courant, who utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain appropriate solutions to vibration system. As the actual model is replaced by a set of finite elements, this method given an approximate solution rather than an exact solution (P.Srinivas, Krishna and Rajesh, 2010). In this study, Finite Element Method will be used to analyses the columns structure.

A higher-order finite element method including geometric non-linearity is addressed in the paper of C.K.Lu and M.A.Bradford (2010) for the analysis of elastic frames for which a single element is used to model each member. The geometric non-linearity in the structure is handled using an updated Lagrangian formulation, which takes the effects of the large translation and rotations that occur at the joints into consideration by accumulating their nodal coordinates. Rigid body movements are eliminated from the local member load displacement relationship for which the total secant stiffness is formulated for evaluating the large member deformations of an element. The influences of the axial force on the member stiffness and the changes in the member chord length are taken into account using a modified bowing function which is formulated in the total secant stiffness relationship, for which that coupling of the axial strain and flexural bowing is included.

To conclude the entire finite element paper and journal that state above, Finite Element Analysis seems like the approximation method to study the strength behaviour of the structure. Finite Element Method gives more accurate result in the global behaviour of the frame system. It is match to may study on frame which uses the finite element analysis to analyses the strength behaviour. As the actual model is replaced by a set of finite elements, this method gives an approximate solution rather than an exact solution. A stiffness matrix from of calculation is one of the calculation methods in finite element analysis.

2.5 ASSUMPTION

There are few assumption need to be made during analysis structure by using the STAAD.Pro software, For a complete analysis, of structure, the necessary matrices are generated on few assumptions such as the coordinate system used in the generation of the required matrices and referred as local and global system. The Structure is idealized into an assembly of beam type element that are joint together through nodes. The assembly is loaded and reacted by applying loads at the nodes and both force and moments may act in any specified direction.

A beam member is a longitudinal structural element having a constant doubly symmetric cross section along the length. Beam members always carries axial forces and may also be subjected to shear and bending on two arbitrary perpendicular planes. This element may also be subjected to torsion. At this point, these beam members are referred as 'member' in the manual.

Internal and external loads acting on each node are in equilibrium. If torsional or bending properties are defined for any structural member, six degree of freedom (DOF) is considered at each node in the generation of relevant matrices. If the members are defined as a truss member, only three degree of freedom is considered in each node.

Local Coordinate axes are assigned to each individual element and oriented so that the computing for element stiffness matrices are generalized and minimized. Global coordinate axes are a common datum for all idealized element so the element forces and displacement may be related to a common frame for reference.

2.6 LOADS ON STRUCTURE

There are many types of loadings that can affect the behaviour of the structures. When designing a building, those loadings should be taken into consideration because if it is miscalculate, absolute failure will occur in the structure.

2.6.1 Static Force

The static forces consist of live load, dead load and forces due to settlement or thermal effect. These types of load are typically being considered when designing every structure.

Live loads are forces that act vertically downward onto the smicture but it is not fixed in character. The value can change anytime and movable. Occupancy and environmental can be described as live loads. Occupancy loads include furniture, stored maternal, human or any other similar material. On the other hand, dead loads are unmovable loads and fixed in behaviour. There are several items that can be mark as dead loads and those are self-weight of the structure, slab, bricks, mechanical equipment or any building properties.

2.6.2 Wind Loads

This two of loading is one or the most important characteristic especially for multi-storey buildings because it can cause sway to the structure. As the wind move, it will deflect or stop when something blocking its path. This phenomenon will cause the kinematic energy transform into potential energy. This Potential energy is called force that induced horizontally onto the structure. Therefore, high-rise building will be design properly to resist this force.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter will discuss about the method that we used to build the model, analysis and design the structure. Finite element method is use to analyse the structure of geodesic dome and its behaviour upon the loading applied. Staad Pro program os used to analyse the strength behaviour of the geodesic dome structure under certain loading. In Staadpro, analyses of the structure are divided into two major parts which are modelling and post processing. In modelling stage, model will built geometry, define the general element material and loading and also define the type of analysis and code used in design. While for post processing stage, the further processing and viewing of the results will see such as the lists of nodal displacement, element force and moments, and deflection plots.

In this study, a model of geodesic dome with 20m diameter and 10 meter height was analyse. Vertical load, horizontal load (wind load) and uniformly distributed load (UDL) was applied to the geodesic dome structure to study the strength behaviour under these conditions. The result show was including the deflection, force on element, stress on element and Utility check for the member used. Value of wind load is referred to Malaysia Standard MS1553:2002 Code of practice on wind loading for building structures.