PROBABILISTICS FINITE ELEMENT ANALYSIS FOR DOUBLE LAYER STEEL SPACE TRUSS

NG CHOON HOW

Report submitted in partial fulfillment of the requirements

for the award of degree of

B.Eng. (Hons.) Civil Engineering

Faculty of Civil Engineering & Earth Resources

UNIVERSITI MALAYSIA PAHANG

JUNE 2014

ABSTRACT

In this thesis, a double layer steel space truss was analyzed by using Probabilistic Method with ANSYS program and validate by Finite Element Method with manual calculation. This analysis is to determine the behaviors of the double layer steel space truss. Besides that, it is also to study of the probabilistic response analysis and input sensitivities of the double layer steel space truss. Nowadays space truss is widely used in many types of construction that included stadium, shopping mall, tower, stage, roof structure and also home design. The difference texture of assembly will definitely beautify a building or a place. In this research structure a 4 x 4 x 0.5 meter for the length, width and height had been used, there have 4 supports at the bottom of the structure, and resist of a total of 16 point loads on the top of the structure. The circular hollow cross section was used for the horizontal member and diagonal member for the structure. A $275N/mm^2$ of yield strength was choose for the steel space truss. The material and geometry of the space truss was satisfying for this research since it passes all the design criteria in designing for tensile, compression and also buckling. The ANSYS results show very good agreement with manual calculation results. From the results of simulation, we get to know the real behavior of the structure under the applied loads. From probabilistic analysis, we got the results of cumulative distribution function, histogram plot, sensitivity plot and simple history plot by 10000 times of simulation. In the simulation results, 5 inputs were assigned to get the probabilistic maximum deflection, axial stress and axial elastic strain. The 5 inputs are force, outer diameter of horizontal member, outer diameter of diagonal member, thickness of horizontal member and thickness of diagonal member. Results from ANSYS program indicated that the maximum deflection is 7.74mm, maximum axial stress is 141.16MPa, and maximum axial elastic strain is 0.6722E-03.

ABSTRAK

Dalam tesis ini, Double layer steel space truss dianalisis dengan menggunakan Kaedah berkebarangkalian dengan program ANSYS dan mengesahkan oleh Finite Elememt Method dengan pengiraan manual. Kajian ini dilakukan untuk menentukan tingkah laku Double layer steel space truss. Selain itu ia juga adalah untuk mengkaji analisis sambutan kebarangkalian dan sensitiviti input Double layer steel space truss. Kini space truss digunakan secara meluas dalam banyak jenis pembinaan yang termasuk stadium, pusat membeli-belah, menara, peringkat, struktur bumbung dan juga reka bentuk rumah. Tekstur perbezaan berhimpun pasti akan mencantikkan bangunan atau tempat. Struktur kajian ini menggunakan 4 x 4 x 0.5 meter sebagai panjang, lebar dan tinggi, selain itu terdapat 4 sokongan di bahagian bawah struktur, dan menentang daripada sejumlah 16 beban titik pada bahagian atas struktur. Keratan rentas berongga bulat digunakan untuk ahli mendatar dan ahli pepenjuru untuk struktur. Double layer steel space truss adalah memilih $275N/mm^2$ sebagai kaji analisis ini. Bahan dan geometri untuk kekuda ruang itu memenuhi kajian ini kerana ia lulus semua kriteria reka bentuk dalam mereka bentuk untuk tegangan, mampatan dan juga lengkokan. Keputusan ANSYS memdapati amat hampir dengan keputusan pengiraan manual. Daripada keputusan simulasi kita mengenali tingkah laku sebenar struktur di bawah beban yang dikenakan. Daripada analisis kebarangkalian, kita mendapat keputusan fungsi kumulatif pengedaran, histogram plot, plot sensitiviti dan sejarah plot mudah oleh 10000 masa simulasi. Dalam keputusan simulasi, 5 input telah ditugaskan untuk mendapatkan pesongan kebarangkalian maksimum, tekanan paksi dan terikan elastik paksi. 5 input adalah berkuat kuasa, diameter luar ahli mendatar, diameter luar anggota pepenjuru, ketebalan ahli mendatar dan ketebalan anggota pepenjuru. Hasil daripada program ANSYS menunjukkan bahawa pesongan maksimum ialah 7.74mm, tekanan paksi maksimum ialah 141.16MPa, dan paksi terikan elastik maksimum ialah 0.6722E-03.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

t	Thickness
f_y	Yield strength
f_u	Ultimate strength
E	Modulus of elasticity
G	Shear modulus
v	Poisson's ratio in elastic stage
α	Imperfection factor
ε	Strain
σ	Stress
γ_{M0}	Partial factor for resistance of cross-sections whatever the class is
γ_{M1}	Partial factor for resistance of members to instability assessed by member checks
γ_{M2}	Partial factor for resistance of cross-sections in tension to fracture
N_{Ed}	Design normal force
N_{Rd}	Design values of the resistance to normal forces
d	Diameter of bolt
χ	Reduction factor for relevant buckling mode
Φ	Value to determine the reduction factor χ
i	Radius of gyration about the relevant axis, determined using the properties of the gross cross-section
λ_1	Slenderness value to determine the relative slenderness
L_{cr}	Buckling length
$[K]$	Global-coordinate structure stiffness matrix

LIST OF ABBREVIATIONS

ANSYS	Analysis System
UMP	Universiti Malaysia Pahang
RHS	Rectangular Hollow Section
CHS	Circular Hollow Section
SHS	Square Hollow Section
2D/3D	Two Dimensional/ Three Dimensional
FEA	Finite Element Analysis
PDS	Probabilistic Design System
MCS	Monte Carlo Simulation
CDF	Cumulative Distribution Function
EC	Eurocode
FZ	Force
OD1	Outer Diameter of Horizontal Member
OD2	Outer Diameter of Diagonal Member
T1	Thickness of Horizontal Member
T2	Thickness of Diagonal Member
EPELAXL	Axial Elastic Strain
SAXL	Axial Stress
DLST	Double Layer Space Truss
LS	Load Step

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND OF THE STUDY

Nowadays space truss is widely used in constructions and also decorated for a structure. Due to it has a nice aesthetic appearance with multi cross over members, it is used for a decorative purpose. The most attractive and famous space truss structure in the world is the Eiffel Tower and the Beijing National Stadium or Nest Bird. According to El-Sheikh (1996), a space truss is commonly used in the large open area with few or no internal supports. A truss in 3-dimensional is always called as a space truss. A simple space truss is in the form of tetrahedrons that joined together by 6 members at the ends (Saeed Moaveni, 1999). A better load distribution and utilization factor will reach with the form of a tetrahedron (Habashi, 2003).

They are some benefits that using space truss as a part of the structure. Prof. S. R. Satish Kumar and Prof. A. R. Santha Kumar state that space truss has a lightweight, structurally efficient and use materials optimally. Besides, the standard size and shape of the space truss can be produced in a factory, thus the labor can assemble handily and quickly at the site. The small size components simplify the operation, carriage and construction. Moreover, by using space truss which can save the space and time for constructing a large column in between the structure. Lastly, the use of complex and costly momentary supports during the construction can be eliminated.

The double layer space trusses that going to analyze by using ANSYS software is as the following. The total surface area of the structure is $16m^2$ with 4m length multiple 4m width. The structure is 0.5m height. There are 3 main components in this structure that are top member, bottom member and diagonal member of the space truss and it will be supported by the 4 supports at the bottom of the structure. The top member is shown as a linear, bottom member is shown as a dash line, and diagonal member is shown as long dash and short dash line whereas the support is shown as a circle. Figure 1.1 shows the dimension and the arrangement of double layer space truss.
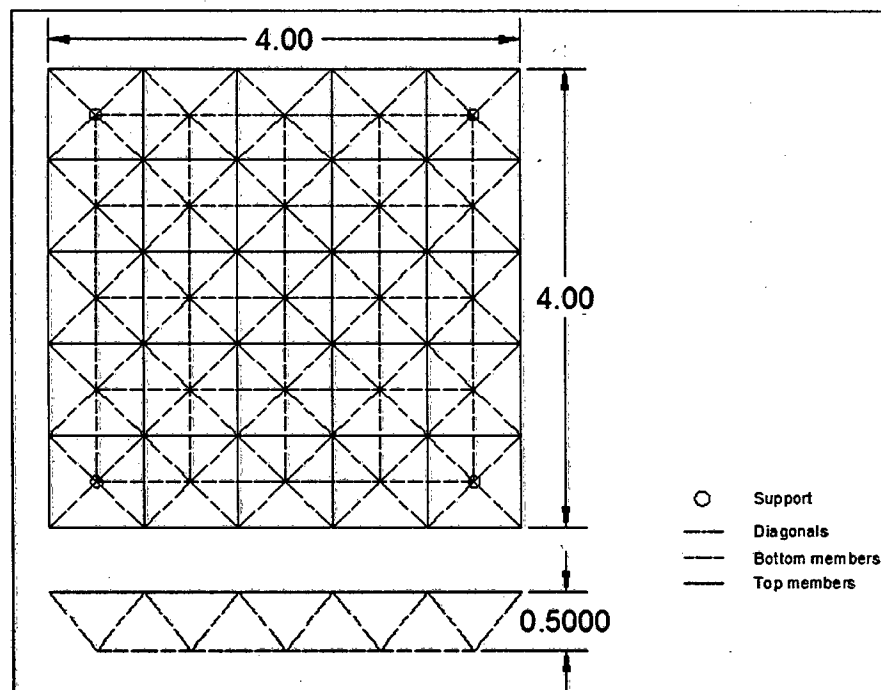

Figure 1.1: Layout of double layer space truss

1.2   STATEMENT OF PROBLEM

Space trusses consist a lot of members that connected together to form a structure. It is complicated to analysis a double layer space truss manually and mistakes are always occurring due to the careless and complicated structure. Time consumes is higher in order

to analysis a structure correctly and accurately. Besides, the cost of the project to design a double layer space truss will be increased due to the time consume is higher. ANSYS is useful to analysis the complicated structure correctly and accurately and it helped to save the cost and time in analyzing a structure. According to Zabihollah (2007), ANSYS is a potent software that using finite element methods to numerically solve a broad variety of mechanical, structural and non-structural problems. The problems may include static or dynamic structural analysis (both linear and nonlinear), fluid, acoustic, electro-magnetic and also heat transfer problems.

Although there are many research on analysis of truss, fame, beam, bridge and tunneling, but there has less research on analysis of double layer steel space truss by using the ANSYS program. Due to unfamiliar used of ANSYS in analyzing the 3D structure. In this research the maximum deflection at the each joint, maximum axial stress and maximum axial elastic strain on each member will be determined by using the ANSYS program.

1.3 RESEARCH OBJECTIVES

Research objectives are an important part for every project or thesis in order to conduct the research well. It acts as guidance for researcher to achieve the final objective and avoid researcher to digress. The main objectives of this research are:

I. To determine the behaviors of double layer steel space truss.
II. To study the probabilistic response analysis of double layer space truss.
III. To study the input sensitivities of the double layer steel space truss.

1.4 SCOPE OF STUDY

This research will be carried out by using ANSYS to analysis the double layer space truss with deflection at each joint, stress and strain at each member. Besides that, develop a

program for analyzing double layer space truss structure. Lastly, conduct probabilistic response analysis on the double layer space truss.

1.5   EXPECTED OUTCOME

This research claims to find out the behaviors of double layer steel space truss. The behaviors are deformation, deflection, axial stress, and axial elastic strain. Besides that it claims to find out the probabilistic response analysis of double layer space truss for all the behaviors. In addition, this research will be carried out to identify the sensitivity input that will affect the most for the double layer space truss. The inputs are force apply, outer diameter of horizontal member, outer diameter of diagonal members, thickness of horizontal member and thickness of diagonal member.

1.6   SIGNIFICANCE OF STUDY

This research will be a significant endeavor in providing information about the steps and results of analyzing the space truss by using ANSYS. It is good to practice analyzing and design the space truss before the construction is started. This study will also be beneficial to the field of civil engineering. This is because the study can provide new information regarding to the space truss. In addition, this research may help to obtain a more economic structure by determining the optimum dimension used. Furthermore, the analyzing of space truss is useful in providing accurate information about the space truss. It can help to prevent the building or structure collapsing due to improper analyzing of it. It will definitely save the life and enhance the reputation of Malaysia. In a nutshell, this study is very significant to the Malaysian and the field of Civil Engineering.

CHAPTER 2

LITERATURE REVIEW

2.1 SPACE TRUSS

2.1.1 Introduction to Space Truss

In general usage of proprietary name or technical literature, a space truss system may call as space frame. Beside that the name of space frame frequently applies to all space grids, including the modular systems that are rightly space trusses. There have an easy way to determine whether it is a 'true' space frame, structure that have no diagonal members are considered space frame (Chilton, 2000). There are a few advantages that using of space grids: load sharing, easy installation of services, robustness, modular components, modular components, regular geometry, and ease of erection. Whereas there are a few disadvantages that using of space grids: high cost, regular geometry, longer erection time, no fire resistance.

Space truss is efficient and light structure type when compared with the traditional beam-column typed structure. Besides that the Capacity of load resistance can be maximized due to it consists both of tension and compression members. Furthermore, the scaffolding and heavy cranes may be eliminated or minimized in using due to the shape of structure procedure is integral with the erection process (C., 2000). With this technique the construction costs will definitely save. It is essential to concern in design and construction.

2.1.2 Types of Space Truss

Space truss càn be categorized into three main types which are simple space truss, compound space truss and complex space truss. The simple space truss is erected in tetrahedron and it can be distended by adding of three members. The compound space truss is the combination between two simple space trusses or more whereas the complex space truss is a truss that cannot be classified as a simple or compound space truss. Figure 2.1, 2.2 and 2.3 show the differences type of space truss.
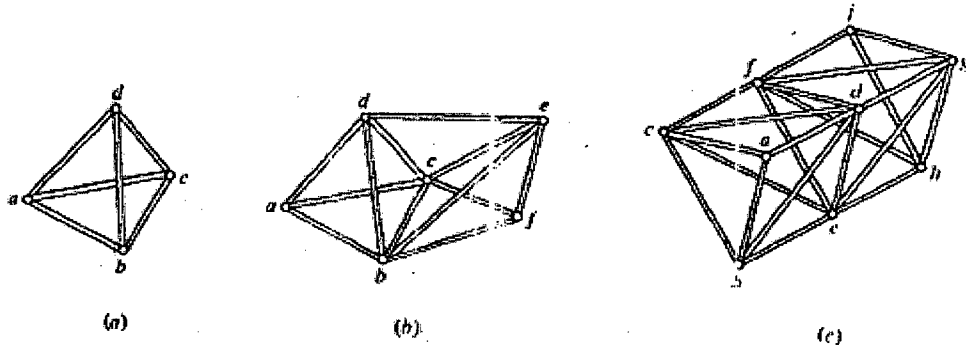

Figure 2.1: Simple space trusses
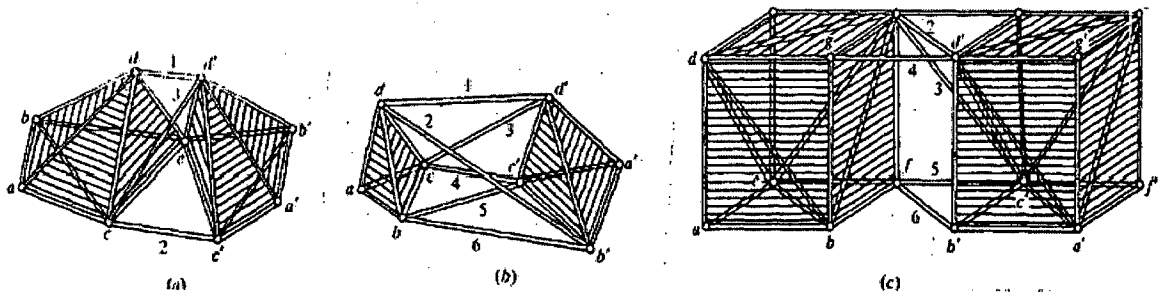
Source: UTM, 2013

Figure 2.2: Compound space trusses
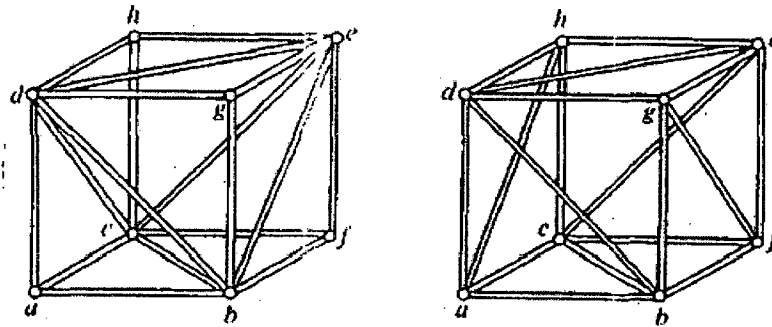
Source: UTM, 2013

Figure 2.3: Complex space trusses

Source: UTM, 2013

2.1.3 Space Tubular truss

The space tubular truss structure has widely used nowadays due to it has an attractive modeling, sustain high load, easy to manufacture, easy assembly. It is widely used in roof structure, stadium, terminal bus station, airport and stage. In space tubular truss structure the axial force generated by bending moment is bearable by the chord member or main tube while the shear force that generated by upper structure is bearable by web member or strut (Dong Shier, 2011).

According to Fen (2005), the stability and resistance of torsion were combined by Square tubes and round tubes due to both of its have the symmetrical and closing section. Round tube and square tube have their own advantage. For round tube the bearing axial force is the advantage while the square tube bending moment is the advantage. The combination of the round tube and square tube had bring a lot of efficiency. The space tubular truss structure has improved the mechanical properties, safety in use and also the consistency of the structures.

2.1.4 Steel Space Truss

Steel space trusses are commonly used of tubular section members. There are quite a few types of links that can be used to connect these members. The most common use to connect the member is the end-flattened connection. There are a lot of advantages of the connections which are cost saving and fast assemblage of the truss. However, there are disadvantages of the connections for example eccentricities and stiffness reduction of members. Whereas the benefits details of steel space trusses that giving by engineers and architects is easy dismantling and transportation for set up in other workplace, more area to accommodate service facilities, mass production, simplicity to expand, fast assemblage, required little or no internal supports in a large open area, light weight and nice appearance, easily incorporate cladding and finishing surfaces (Luciano Mendes Bezerra, 2009).

2.1.5 Types of Section

According to El-Sheikh (1999), there are three types of section Rectangular hollow section (RHS), Flat bars, and Circular hollow section (CHS) that use in the Catrus truss. The main compression forces were resisted by RHS in the top chord. Besides the tension forces were resisted by flat bars in the bottom chord. However, RHS members should be used in the bottom chord instead of the flat bars where the bottom members are going to receive any of the compression forces. The CHS is suggested to use in the diagonal members due to it has an adequate behavior under both tension and compression.

From BS EN 1993-1-1:2005 there are several shapes and dimension of hot-finish steel that use in the market nowadays. The most common used is rectangular hollow section (RHS), circular hollow section (CHS) and square hollow section (SHS). RHS and SHS describe by their depth, width and wall thickness. An example is a 40 x 20 x 2 RHS, which is 40 mm deep, 20 mm wide and the wall is 2 mm thick. However the CHS describes by their diameter and wall thickness. Figure 2.4 shows the circular hollow sections.

BS EN 1993-1-1:2005
BS EN 10210-2:2006

**HOT-FINISHED
CIRCULAR HOLLOW SECTIONS**

Celsius® CHS
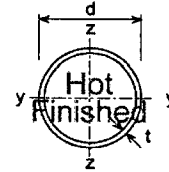
Dimensions and properties

Section Designation		Mass per Metre	Area of Section	Ratio for Local Buckling	Second Moment of Area	Radius of Gyration	Elastic Modulus	Plastic Modulus	Torsional Constants		Surface Area	
Outside Diameter d mm	Thickness t mm	kg/m	A cm²	d/t	I cm⁴	i cm	W_{el} cm³	W_{pl} cm³	i_T cm⁴	W_t cm³	Per Metre m²	Per Tonne m²
219.1	5.0	26.4	33.6	43.8	1930	7.57	176	229	3860	352	0.688	26.1
	6.3	33.1	42.1	34.8	2390	7.53	218	285	4770	436	0.688	20.8
	8.0	41.6	53.1	27.4	2960	7.47	270	357	5920	540	0.688	16.5
	10.0	51.6	65.7	21.9	3600	7.40	328	438	7200	657	0.688	13.3
	12.5	63.7	81.1	17.5	4350	7.32	397	534	8690	793	0.688	10.8
	14.2	71.8	91.4	15.4	4820	7.26	440	597	9640	880	0.688	9.59
	16.0	80.1	102	13.7	5300	7.20	483	661	10600	967	0.688	8.59
244.5	8.0	46.7	59.4	30.6	4160	8.37	340	448	8320	681	0.768	16.5
	10.0	57.8	73.7	24.5	5070	8.30	415	550	10100	830	0.768	13.3
	12.5	71.5	91.1	19.6	6150	8.21	503	673	12300	1010	0.768	10.7
	14.2	80.6	103	17.2	6840	8.16	559	754	13700	1120	0.768	9.52
	16.0	90.2	115	15.3	7530	8.10	616	837	15100	1230	0.768	8.52

Figure 2.4: Circular hollow sections

Source: BS EN 1993-1-1:2005

2.1.6 Connection

In general, truss members are connected each other together with pins or fix method. Weld or bolt connections are preferred due to fast assemble, costs saving, easy dismantling, and employee availability. Beside that bolted connections are usually suggested for medium and large span trusses. There are examples of steel trusses that connected only by bolts and it is able to cover the spans up to 60m (Luciano Mendes Bezerra, 2009).

There are some jointed type connections that used to connect the truss member which are bolted and screws, or multiply pressed. According to Eurocode 3, the famous connection for truss members are bolting and welding. The bolt size and weld length depends on the values of axial force in the member. Compression and tension member have the different design in the connection.

2.2 ANSYS PROGRAM

2.2.1 Introduction of ANSYS

ANSYS is a common purpose finite element modeling package for numerically solving a large variety of mechanical problems. These problems include of static and dynamic structural analysis, heat transfer and fluid problems, as well as acoustic and electro-magnetic problems (ANSYS Tutorials, 2001). In general, ANSYS finite element analysis has 3 fundamental steps that consist of preprocessing phase, solution phase and post processing phase.

For preprocessing phase, finite element is form and detached from the solution area. That is split the problem into nodes and elements. Then, suppose a structure function to clarify the physical behavior of a component. Secondary phase of solution phase is to get the nodal results such as the displacement values and temperature value at different nodes by solving a set of linear and nonlinear algebraic equations simultaneously. The postprocessing phase is to gain other significant results. (Saeed Moaveni, 1999).

2.2.2 Element Types

In ANSYS program, there are more than 100 types of element which use to analysis different type of problem. Beside that the element type was identified by category name follow by a number. Selecting of the element correctly is very important in the analysis of the model structure. There are some elements are to analysis the structure in the 2D/3D form while some of the elements are to analysis the structure with non-linear/linear. For example the name of PLANE was represented the two dimensional solid elements (Moaveni, 1999). For the truss element there are a few element types that can be used. That is Link8, Link 10 and Link180. In 3D structure Link10 and Link180 are suggested to use. Figure 2.5 shows the differences type of elements.

Structural Point	Structural 2-D Line	Structural 2-D Beam		
Structural Mass ●	Spar	Elastic Beam	Plastic Beam	Offset Tapered Unsymmetric Beam
MASS21 1 node 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	LINK1 2 nodes 2-D space DOF: UX, UY	BEAM3 2 nodes 2-D space DOF: UX, UY, ROTZ	BEAM23 2 nodes 2-D space DOF: UX, UY, ROTZ	BEAM54 2 nodes 2-D space DOF: UX, UY, ROTZ
Structural 3-D Line			Structural 3-D Beam	
Spar	Tension-Only Spar	Linear Actuator	Elastic Beam	Thin Walled Plastic Beam
LINK8 2 nodes 3-D space DOF: UX, UY, UZ	LINK10 2 nodes 3-D space DOF: UX, UY, UZ	LINK11 2 nodes 3-D space DOF: UX, UY, UZ	BEAM4 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ	BEAM24 2 nodes 3-D space DOF: UX, UY, UZ, ROTX, ROTY, ROTZ

Figure 2.5: Differences type of elements

Source: Moaveni, 1999

2.3   ANALYSIS METHODS

2.3.1   Probabilistic Analysis Method

To avoid structural failure during or after the construction, structural engineer needs to predict static displacement and stress response of a structure. Due to the uncertain effects of the structural materials, physical parameter and geometry dimensions get on uncertainties consequentially (Juan Ma, 2006). However the probabilistic methods cannot give an accurate result except there are a few experimental data information that are available to validate the assumption of the probability densities of the random variable. If the standard deviation cannot be got, then the probabilistic analysis method is not available.

Besides that, in several circumstances we only can be able to get the range of the structural parameter with lower and upper bounds. Therefore the interval analysis method will be very useful due to the lower and upper bounds are required for the interval analysis method only. A bounded uncertain parameter can be described as an interval variable (Wei Gao, 2008). Tie Ming (2011) stated that the probabilistic analysis approach was applied to

account for the variability in fabrication. Probabilistic method is another ways to analysis a structure to achieve a strong and reliable design efficiencies by using finite element modeling. From the analysis results, it found that probabilistic analysis is closer to reality when compared with the deterministic methods.

2.3.2 Monte-Carlo Simulation Method

According to Kralik (2009), Monte Carlo methods are based on a simulation of the input stochastic parameters according to the expected probability distribution. This method has the pros and cons. The advantages of this method are the last values of the reliability reserves can be continuously showed in the form of a histogram or cumulative function; the simulations are independent, easy to understand and clear, and enables the estimation of the statistical discrepancy of the estimation on a particular relevance level. The disadvantages of this method are a huge number of simulations for small probability values, and taking a long time for calculation of complex problems.

According to Gao (2007), Monte-Carlo simulation method and perturbation method are generally used for the static and dynamic analysis of structures with uncertainties. In the Monte-Carlo method, the structural parameter value varies within a given range. The dynamic analyze on the same structure is performed in a huge amount. Besides the natural frequencies and mode shapes are found through the statistical data of the mean value and standard deviation. However to obtain the dynamic characteristics of random structures by using Monte-Carlo simulation method, a large amount of computational work is needed.

2.3.3 Finite Element Analysis

The main purpose of finite element analysis (FEA) is to present a suitable numerical model that accurately calculates such mechanical behaviors as deflection and strain (Dongdong Zhang, 2014). The finite-element analysis can provide a series of deformed shapes from the initial one to the ultimate deformed shape required of the planar layout. By using the finite-element analysis the posttensioning forces and induced stresses can be

found from the current results once the final space shape is determined (Hewen Li and Lewis C. Schmidt, 1997). The most general accomplishment of the finite element method, the direct stiffness approach is used. With this method, a set of simpler, idealized elements interconnected at the nodes must be in the system. By solving a set of equations the unknown displacement and forces in the structure can then be determined.

2.3.4 Sensitivity Analysis

Sensitivity analysis with show the influence percentage of the variable input parameters to the reliability of the structures in bar chart and also pie chart. Matrix of the correlation coefficients of the input and output parameters is defined by Spearman. Figure 2.6 shows the results of the sensitivity analysis of the interstorey drift and the shear forces in models RSM2 in the case of the seismic action. The sensitivity analysis provides the important information about the influence of the uncertainties of input variables (load, material, models) to engineer for optimal design of the structures (Kralik, 2009).
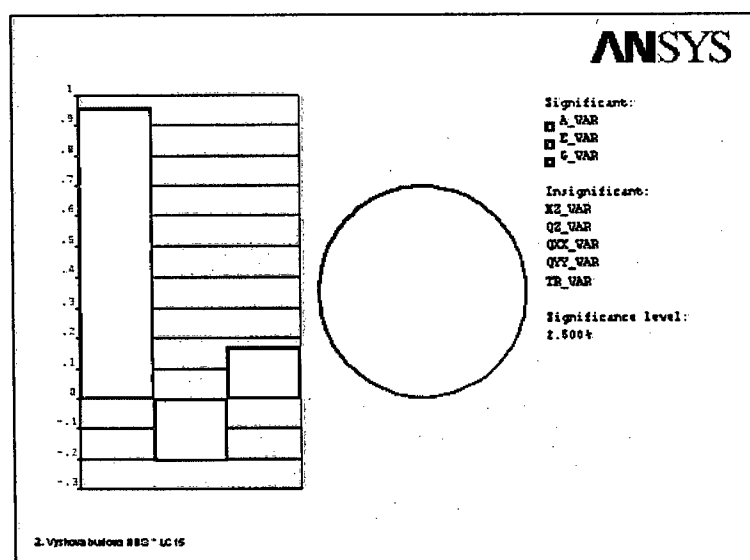

Figure 2.6: The results from the sensitivity analysis of the Dmax and Vmax in the model RSM2

Source: Kralik, 2009