Response spectrum analysis of medium rise buildings subjected To a far-field earthquake

By

MOHD SHAHRIN B AWALI

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

JANUARY 2008

Universiti Teknologi Petronas
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Response spectrum analysis of medium rise building subjected to far-field earthquakes

By

Mohd Shahrin Bin Awali (5961)

A project dissertation submitted to the
Civil Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(CIVIL ENGINEERING)

Approved by,

(Dr. Victor Rivera Macam Jr)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

January 2008

CERTIFICATION OF ORIGINALITY

This is to certify that I 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.

MOHD SHAHRIN BIN AWALI

ABSTRACT

Normally in Malaysia, lateral force that is taken into consideration in design is wind load. Tremors felt by Malaysians from the long distance Sumatran Earthquake on the 26^{th} December 2004 was a "wake up call" for engineers, architects, local authorities and etc to accept the importance of understanding potential seismic hazard that can occur to building and others structure.

Seismic loading requires an understanding of the structural behavior under large inelastic, cyclic deformation due to inertial forces of the building mass induced by the shaking of its foundation. The main problem caused by earthquake to building is vibration. Thus, the damage on building is not caused by seismic ground motion but internally generated inertial forces caused by vibration of the building mass.

Based on the simulation done using Staad-Pro software on standard school building design in Malaysia due to Response Spectrum Acceleration of far-field earthquake, this research conclude that our school building is still safe under far-field earthquake loading.

ACKNOWLEDGEMENT

This project would not have been possible without the assistance and guidance of certain individuals and organization whose contributions have helped in its completion.

First and foremost, thank god for His grace and mercy that He gave to me, without it, maybe I cannot make it through until the end. I would like to extend my most sincere gratitude to my project Supervisor, Dr Victor Rivera Macam Jr. for his full support and continuous encouragement and guidance for me towards completing this Final Year Project (FYP), and also thank you for monitoring my progress and for helping me to face any problem related to my project. Almost certainly without his cooperation, I might not be able to fulfill the project requirements, as it needs a lot of commitment towards finishing it.

I am greatly indebted to my parent, for their moral support and inspiration over this project period. I also would like to express my gratitude to Mrs. Salmia Beddu from UTP for her non-stop commitment in giving me support and guidance.

My acknowledgement would be incomplete without giving credit to UTP, especially Civil Engineering Department who has equipped students with essential skills for self-learning. A word of sincere gratitude to the Final Year Project committee for arranging various seminars as support and knowledge to assist me. The seminars were indeed very helpful and insightful. Special thanks to all lecturers from Universiti Teknologi Petronas who had provided untiring guidance throughout the period of the project.

Last, but certainly not least, I would like to apologize if any party was inadvertently excluded from being mentioned above and would like to thank all parties that were involved in making this project a success.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.0 BACKGROUND

The Indian Ocean earthquake, known by the scientific community as the Sumatra-Andaman earthquake on December 2004 which caused the deadly tsunami was felt in Malaysia, though at a small scale which didn't cause major damages such as collapsed buildings with casualties. But from then, the issue of whether buildings in Malaysia are able to withstand the shaking if the same magnitude of earthquake were to happen somewhere nearer to Malaysia rises and people starts to express their concern on multistory and high-rise buildings safety which they have been using or living in.

Tremors felt by Malaysians from recent earthquakes in neighboring country have made the people worried about their safety while occupying multistory buildings. Identification of the weakest elements of the existing reinforced concrete structure were necessary for further actions to be taken, in order to reduce the number of structural and non-structural damages, casualties, loss of live and others.

The growth in modern medium rise building construction has been largely for commercial and residential purpose in Malaysia due to the rapid economic growth and industrialization. Multi-storey buildings are uniquely characterized by requiring that lateral loads be a major design consideration. Two types of loads which are normally associated with lateral loads are wind and earthquake load.

However, this study will be focused on typical school buildings. A three storey building based on JKR drawings that has been used since 1991, is referred to analyze the dynamic load capacity. School building is chose because it is an important multistory building which its safety has always been emphasized.

1.1 PROBLEM STATEMENT

Currently, the researches on the effect of wind load and seismic load on multi-storey building have been carried out through experimental work and computer modeling. Nowadays, this is become a hot topic in Malaysia for further study since the incident of Tsunami on end of year 2004 and the worst storm due to tropical cyclone. Therefore, the lateral load effect on the medium rise building is getting more attention other than high rise building.

School buildings in Malaysia have never been design to withstand earthquake. In reality, Peninsular Malaysia is located relatively far away from Sumatran seismic zones and has never seen any damage before, but rapid construction of high rise structures in Peninsular Malaysia may create high seismic risk in terms of structures damages, loss of lives and assets due to high concentration of population and commercial activities taking place in the structure.

Event most of the software structure in design analysis required seismic force or input such as time history or design response spectrum, but in Malaysia, it does not have any provision for earthquake load in designing building structures. Most of the existing reinforced concrete structures in Malaysia are largely designed according to BS 8110 on the combination of gravity and wind load and does not have any provision on seismic load. The safety of the school building when an earthquake strike can be questioned especially when hundreds of peoples are in it at the time of occurrence.

1.2 OBJECTIVES OF REPORT

The objectives of the project are as follows:

1. To study the behavior of structure under dynamic loading.
2. To determine the moment, deflection and shear of the building due to far-field earthquake load by using response spectrum acceleration.
3. To determine either the building is safe under earthquake loading.
4. To analyze the building structural performance using STAAD Pro.

1.3 SCOPE OF STUDY

This study covers in the following aspects:

a. To analyze the standard school building design by using STAADPro 2005 software.

b. All plan dimensions such as length and width of the building taken from standard JKR design.

c. All characteristic strength of materials such as concrete grade and reinforcement details taken from standard JKR drawing.

d. The analysis and design exercises are limited to linear elastic analysis.

e. All concrete designs are referred to BS 8110: Part 1: 1997.

f. All live loads desired in the analysis are referred to BS6399: Part 1: 1996

The scope of this study would be on dynamic loading by analysis and modeling. For this paper, it would focus only on the analysis. The initial works are to calculate flexural/bending capacity, shear capacity, and cross section of properties and moment of inertia of beams as present in the JKR drawing. From there, a model can be simulated and the experiment of how the building behaves under seismic actions of the structure can be done. Some research was done to find and summarize every journals or articles related to the topic. This is important as any studies regarding influence of seismic on school buildings in Malaysia have ever being done. Every research concerning to it will be studied and the results would be included in this paper.

The benefits that would be obtained at the end of this research would definitely be helpful to people as they would finally know if the school building would be safe or not if another major earthquake would to happen in the future.

CHAPTER 2
LITERATURE REVIEW

2.1 Introduction

Public building structures in Malaysia include hospitals, schools, offices; quarters have been heavily developed for many states in the country. The performance of the structures against seismic hazard effects human safety, loss of properties and maintenance cost. Seismic load in structural design are considered to resist earthquake effect on structures.

Lateral loads due to wind or earthquake are the major factors to be considered in design, especially for high-rise buildings. According to Smith and Coull (1991), for building up to 10 stories and of typical proportions, the design is rarely infected by wind loads. Great Sumatran-Andaman earthquake disaster occurred on 26 December 2004, becomes an important issue in Malaysia especially on the effect of the existing RC structure due to earthquake. Identification of the weakest elements of the existing reinforced concrete structure were necessary for further actions to be taken, in order to reduce the number of structural and non-structural damages, casualties, loss of live and others.

2.2 Lateral load

Most of the lateral loads are live loads whose main component is lateral force acting on the building structure. Luebkeman and Peting (1996) reported that typical lateral loads would be a wind load against a facade, an earthquake, the earth pressure against a beach front retaining wall or the earth pressure against a basement wall. Most of the horizontal loads vary in intensity depending on the building's geographic location, structural materials, height and shape.

2.3 Seismic load

Petersen et al., 2004 have developed or modified earthquake catalogs for Malaysia to include only independents earthquakes. These catalogs were used to define four source zones that characterize earthquake in four tectonic environments: subduction zone interface earthquakes, subduction zone deep intraslab earthquake, strike-slip transform earthquake, and intraplate earthquakes. Force distance beyond 200km (Malaysia), the applied ground motion was predicted from California (Interplate) and India (Intraplate) strong motion data.

Several past years the research for Malaysia has been conducted by the Structural Earthquake Engineering Research (SEER) group from Universiti Teknologi Malaysia (UTM) on seismic analysis to obtain Peak Ground Acceleration (PGA) and Spectral Acceleration (SPA) at bedrock as the effect of the earthquake to Penang and Kuala Lumpur. The result of the analysis showed that the PGA's at bedrock for Penang in accordance with attenuation relationships from Youngs and Atkinson and Boore are 3.89 gal and 4.84 gal. The effect of the earthquake in Kuala Lumpur is lower about 25% to 31% than Penang. According to the attenuation relationships from Young, Atkinson and Boore , the PGA at bedrock for Kuala Lumpur is 2.90 gal and 3.32 gal (Adnan. A, 2002).

Megawati et al., (2003), had developed a new set of attenuation relationship for shallow crustal earthquake in stable continent and active tectonic region for Singapore and the Malays Peninsula since the number of recorded ground motions in the region is very limited.. From this study, the Sumatran Fault Segments have the potential to generate a specified level of response spectral acceleration in Singapore and Kuala Lumpur based on the newly derived ground motion models.

2.4 Load combinations

Basically, building systems are subjected to vertical loads due to dead and live loads acting downwards. Besides of these load, the building is subjected to lateral loads mainly resulting from wind forces. It is usually assumed that these lateral loads acting along different directions in two mutually orthogonal and reversible directions.

Methods of combining types of loadings vary according to the design method and the Code of Practice concerned. Although dead load is considered to act in full all the time, but imposed loads and lateral loads do not necessarily do so. Besides that, the probability of the full gravity live loading acting with either the full wind, earthquake or temperature loadings is low and of all of them acting together is even lower. Furthermore, wind and earthquakes are assumed never to act simultaneously. (Smith and Coull, 1991)

2.5 Past research

i. **Yong Lu (2002)** conducted a study on seismic behavior of multistory RC framed structures. Under the Bare Frame RC model, it was designed to satisfy seismic requirements for ductility class "Medium" with design peak ground acceleration (PGA) of 0.30g. Models are reduced to 1:5.5 and tested under simulated earthquake on an earthquake simulator. Cracking pattern and failure modes were observed. At 0.3g, a uniformly distributed cracking pattern appeared. At the next test of 0.9g, the general response Bare frame RC remained stable, but the cracking widened substantially (maximum crack width on the model exceeded 1mm) indicating excessive yielding, while spalling of concrete occurred in lower stories. Plastic hinges appeared to also occur in several columns, due to "relaxation" of column design moments from satisfying equilibrium around joints. The diagonal cracks that occurred at some beam-column joints were generally light but due to heavy reinforcements at the joints, the cracks became critical. Yong Lu also discovered there was severe damage at place where there is abrupt reduction (over 20%) of the column cross section size. The phenomenon was caused by an intensified high mode "whipping "effects.

ii. **Laura N Lowes and Arash Altoontash (2003)** describes that, under gravity and earthquake loading, beams would develop nominal flexural strength while column longitudinal reinforcement would carry tensile stresses that approach the yield strength. In their study, the frame member loads act as tension and carried by frame member longitudinal steel and concrete, while shear is carried by concrete. The load carried in frame member concrete meanwhile was transferred directly into the joint core concrete. The joint strength is dependent on bond strength and the joint stiffness depends on anchorage-zone deformation. Depending on the bond stress distribution, joint shear may be distributed uniformly or maybe transferred primarily through a diagonal compression strut that develops within joint core. Joint stiffness and strength determined by the response of the joint core under nominal shear loading. Shear is transferred from frame members into the joint at the perimeter of the joint and is assumed to occur across closed concrete cracks in the vicinity of frame member flexural compression zones. If cracks at the perimeter of the joint remain open under load reversal, strength and stiffness of the interface shear transfer mechanism is reduced.

iii. It is common for columns in multistory frame structures subjected to lateral loading to be under a shear dominant loading condition with reversed bending along the column height. **Yan Xiao and Armen Martirossyan** (1998) observed that up to certain displacement ductility levels, flexural cracks perpendicular to the column axis developed first in regions close to the top and bottom ends of the columns. The flexural cracks became inclined and extended into the web zone of the columns due to influence of shear, at the stage exceeding the first yield of longitudinal bars. At later stages of loading, independent cracks started to occur and plastic hinges were fully formed at top and bottom of columns. In terms of flexural failure, the columns lose their capacities at some point due to longitudinal bar buckling accompanied by crushing of concrete. The ultimate performance for columns with smaller longitudinal bars, subjected to low axial load and high axial load are dominated by load-carrying capacity degradation upon cycling due to buckling of compression longitudinal bars within the columns plastic hinges. Differences in transverse steel content did not effect significant changes in the maximum load-carrying capacities. Maximum capacities were typically reached corresponding to the crushing of cover of concrete and at that stage; the transverse reinforcement was not fully activated.

However, ultimate deformations are reduced for columns with less transverse reinforcements, particularly for columns subjected to axial load equal to 20% of the column axial load capacity. The increase in smaller transverse reinforcement strains from the experiment indicates the degradation of concrete shear contributions at large displacement ductility levels. Also, columns with larger longitudinal steel content were found to developed higher lateral loading. Increased in flexural capacity also implied an increased in shear demand. However, despite the increase in shear demands, bigger longitudinal bars demonstrated improved ductility compared with columns with smaller longitudinal bars.

iv. In the dynamic time-history analysis, actual recorded motions were used. All motions include a longitudinal and a transverse component. **Andreas J. Kappos and Georgios Panagopoulos (2004)** improves it by scaling the intensity of the design spectrum using a modified Housner technique based on the area under the pseudo velocity spectrum, in the range from 0.8*T0* to 1.2*T0*, to significantly reduce scatter in the calculated response. In the analysis, the post yield behavior was modeled using bi-linear *M-θ* curves for each principal direction, with no account for *Mx-My-N* interaction.

A non-linear static pushover analysis is a simple option for estimating the strength capacity in the post elastic range. This procedure involves applying a predefined lateral load pattern that is distributed along building height. The lateral forces are then monotonically increased in constant proportion with a displacement control in the top of building, until a certain level of deformation is achieved. The method allows tracing the sequence yielding and failure of structural member, as well as the progress of overall capacity curve of the structure. The pushover analysis is carried out in two sequential stages: a first stage involving the application of the gravity loads; and a second stage in which the lateral loads were incrementally applied. The application of the gravity loads was performed in one single step due to the fact that none of the elements reached its yield or cracking strength.

v. A pushover analysis model also requires a series of hinges to account for the nonlinear behavior of the various structural elements which include RC beams, columns and the masonry elements. Another important aspect in pushover analyses also, is the definition of the lateral loads. In **J. Proença, Carlos S. Oliveira and J.P. Almeida (2002)** studies, the lateral loads were considered proportional to the product of the storey masses and the fundamental elastic mode shape. It was assumed that the mode accurately describes the predominant response pattern of the structure.

Here the author are trying to do a new research that never been done by any other research base on title Response spectrum analysis on medium rise building subjected to far field earthquake. The author hopes that this research will give benefits to the authorities and construction industries and make sure the building is safe to occupy.

CHAPTER 3

METHODOLOGY / PROJECT WORK

3.1 Introduction

This project is to study the behaviors of medium rise building subjected to far-field earthquake. In order to analyze the seismic performance of the buildings, a single main frame was chosen for modeling in finite element analysis. The building model is from the standard school design by Jabatan Kerja Raya (JKR) Malaysia assumed to be located at Ipoh. Table 3.1 shows the building data in detail. In this study, the existing building dimension is remained the same. The analyses of the structure are using Staad-Pro 2004 software. Figure 3.2 shows the flow chart of the analysis.

Table 3.1: Building Data

No.	Remark	Data
1.	Location	Ipoh, Perak
2.	Terrain	Area with no obstruction
3.	Height of Storey	3.0 m
4.	Height of building	9.65 m
5.	Width (plan)	7.5 m
6.	Building Usage	School
7.	Materials	Reinforced Concrete
8.	Grade of Concrete	30
9	Concrete Density	24 kN/m3
10	Exposure Conditions	Moderate
11	Fire Resistance	2 hours

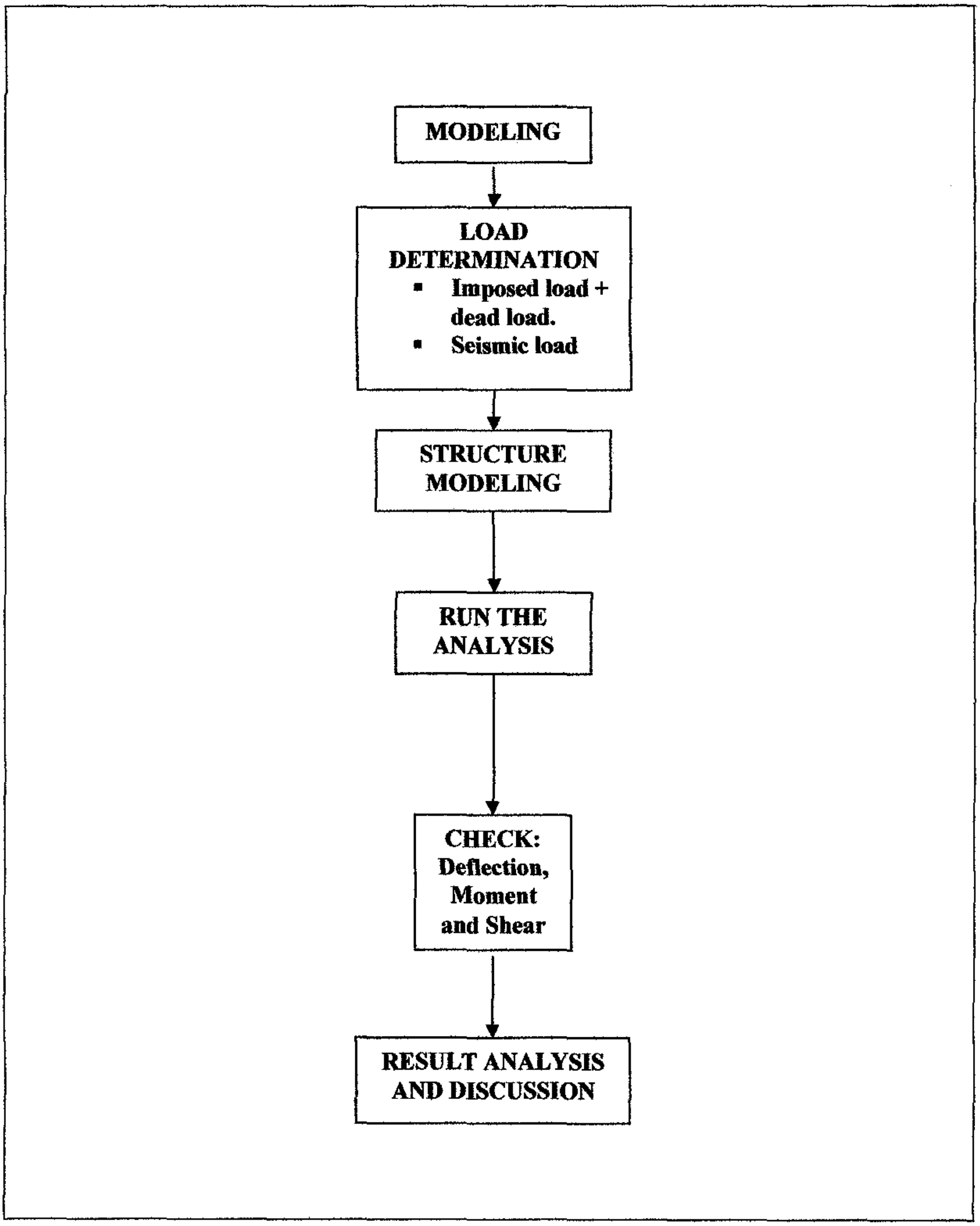


Figure 3.2: Flow Chart of Analysis

Member/Element	Sizes (mm)
Columns	250x350
Beams	200x550
Roof Beams	200x400
Slabs	175 (thickness)

Table 3.3: Dimension for columns, beams and slab

3.2 Load Determination

The load specified in this study is suitable for use either with strength design or with allowable stress design criteria. The load consists of dead load; G_k, live load; Q_k and dynamic load. Three load combinations have been considered.

3.2.1 Gravity Load

i. Dead Load

In this study, dead loads used consist of:

a) Self-weight = 1 kN/m^2

b) Brick loads = (3.0-0.5) x 3 kN/m^2

= 7.5 kN/m

where 0.5 m is the thickness of beam and 3.0 m is the inter-storey height of building.

c) Finishes = 1.2 kN/m^2

ii. Live Load

Live loads are loads of varying magnitudes and / or positions caused by the use of the structure. Live loads for buildings are usually specified as uniformly distributed surface loads in kN per unit area.

d) Imposed load (floor) = 1.5 kN/m^2

3.2.2 Seismic Design

During the earthquake, the ground surface moves in all directions. The movements parallel to the ground surface generally cause the most damaging effects on stationary structures, because structures are ordinarily designed to support vertical gravity loads (Ambrose and Vergun, *1999*). Seismic load in the form of response spectrum is applied in this analysis for the purpose of this study. The response spectrum is taken from **Rashwan et. al (2007).**

The response spectrum analysis is the preferred method in this study because it easier to use. The time-history procedure is used of it is important to represent inelastic response characteristic or to incorporate time-dependent effects when computing the structure's dynamic response (Taranath, 2005).

A response spectrum is simply a plot of the peak (displacement, velocity or acceleration) of a series of oscillators of varying natural frequency, which are forced into motion by the same base vibration. The resulting plot can then be used to pick off the response of any linear system, given its natural frequency of oscillation. The response of a structure to an earthquake may refer to stress, displacement, acceleration, velocity, shear or any other parameter affected by ground motion.

In this study, the building is considered located on a very dense soil and soft rock (soil class C). The design spectra acceleration and the time period are shown in figure 3.6 and Table 3.5, respectively.

Site Class or Soil Profile Type	Description	Shear Wave Velocity, $\overline{V}_{30}$ Top 30 m (m/sec)
A	Hard Rock	>1500
B	Rock	760 – 1500
C	Very dense soil/soft rock	360 – 760
D	Stiff soil	180 – 360
E	Soft Soil	<180

Table 3.4 Soil Class based on NEHRP 9

Table 3.5 Time Period for soil class C (Rashwan et al, 2007)

Period	Modified
0.01	0.0988
0.09	0.2470
0.47	0.2470
0.80	0.1463
1.00	0.1170
1.50	0.0780
2.00	0.0585
2.50	0.0468
3.00	0.0390
3.50	0.0334
4.00	0.0293
4.50	0.0260
5.00	0.0234
5.50	0.0213
6.00	0.0195
6.50	0.0180
7.00	0.0167
7.50	0.0156
8.00	0.0146
8.50	0.0138
9.00	0.0130
9.50	0.0123
10.00	0.0117

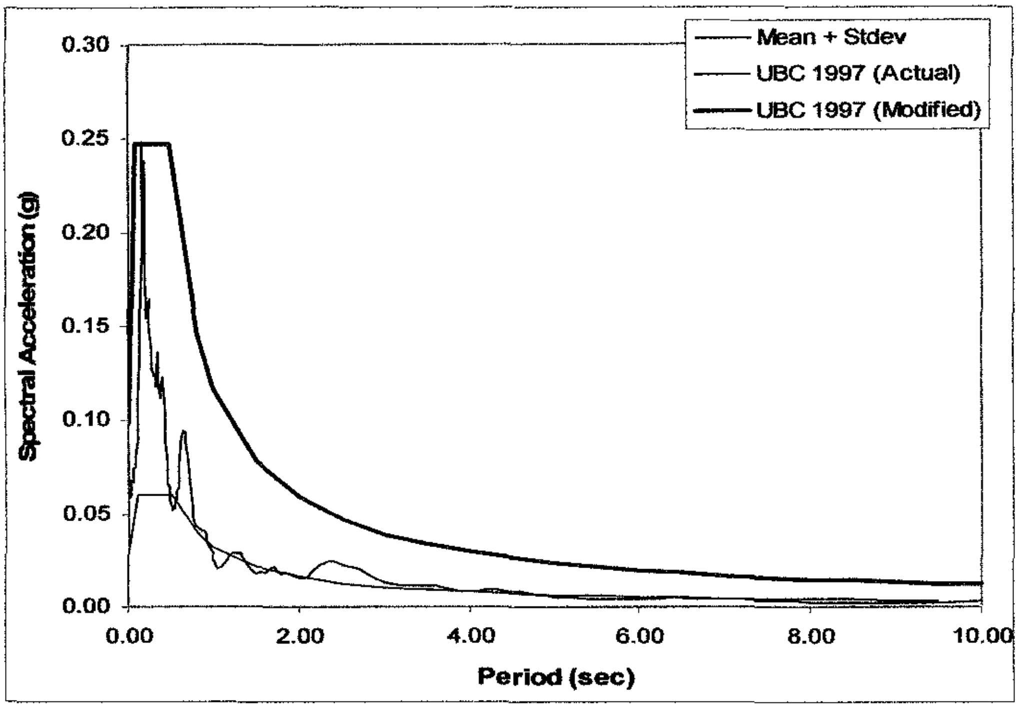


Figure 3.6 Design spectra acceleration for soil class C (Rashwan et al, 2007)

3.3 Load Combination

The load combination used in this study is based on Uniform Building Code (UBC) 1997 and British Standard BS 8110. Ultimate Load Combinations for concrete structure are as follows:

i) U = 1.4DL + 1.6LL

ii) U = 1.4DL + 1.7LL

iii) U = 0.9DL + 1.0EQ

Where:

U = ultimate load resulting from load combinations

DL = dead load

LL = live load

EQ = Earthquake load

3.4 STAAD Pro 2004 Structural Analysis Software

STAAD.Pro is a general purpose structural analysis and design program with applications primarily in the building industry - commercial buildings, bridges and highway structures, industrial structures, chemical plant structures, dams, retaining walls, turbine foundations, culverts and other embedded structures, etc.

STAADPro (Structural Analysis and Design) is a comprehensive structural engineering program capable of performing analyses of three-dimensional structures. In this study, STAADPro 2004 computer software is used to analyze the models of the wall-frame structures.

Design and analysis carried out by STAADPro 2004 are divided into 3 phases which are pre-processing (modeling), analysis and post-processing. The analysis carried out in this modeling is based on finite element analysis. The user communicates with STAADPro 2004 through an input file editor. This input file can be generated either graphically or by typing simple English language based commands.

Firstly, the building model is defined as a SPACE structure type which is consisting of beam and column members and plate elements. A Space type is one where the structure, the loading or both, cause the structure to deform in all 3 global axes (X, Y and Z). It is a three dimensional structure where the loads are applied in any planes. Afterward, user has to generate the model geometry which consists of joint members, their coordinates, member numbers, the member connectivity information, plate element numbers, etc.

In general, the term MEMBER will be used to define the frame elements like beam and column whereas the term ELEMENT will be used to define the plate elements such as shear wall and slab. In the thesis, the Conventional Cartesian Coordinate System is used to define the structure geometries and load patterns. This coordinate system is follow orthogonal right hand rule.

Then, assigning the member and element properties of that structures following by the material constants such as modulus of elasticity, weight density, Poisson's ratio and so on. Caution has to be given to the force and length units to facilitate the input data. After that, support types are specified as FIXED which restraints against all the directions of movement at the base level.

Afterwards, the primary load cases have to be created for this structure which includes dead load, live load and seismic load. Currently, loadings in this structural program can be specified as joint load, member load and element load. After that, generate the self-weight of the structure in STAADPro following by the self-weight of the brick wall as member loads and slab finishing as an element load. All of them are used as uniformly distributed loads in analysis and define as Load 1.

After that, the live load for this structure is assigned as a pressure loads acting uniformly on the slab and defined as Load 2. The live load for the office general use is based on BS 6399 Part1: 1996.

PERFORM ANALYSIS is used to specify the analysis purpose. The StaadPro run is terminated using the FINISH command as a last input command to end processing of the file. This is useful while troubleshooting for input data errors. User has to perform the analysis and design to the model in order to obtain the displacements, forces, stresses and reactions in the structure due to applied loads.

Design phase is involved once the pass-fail status of the members and elements for the requirement of reinforced concrete code (BS 8110) is to be determined. If there are no errors in the input, the analysis is successfully completed.

Finally, user has to run the post-processing mode in order to view the results and assess the suitability of the structure from the standpoint of safety, serviceability and efficiency. Besides that, user can also create the customized reports from the post-processing phase.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

Medium-rise building modeling and analysis by using STAADPro 2005 is easier than using manual method. Values of shear, bending moment, deflection, reinforcement and member size requirements used for each member can be displayed at the output of the program. By referring to the main objective of this study, the important parameters to be determined are deflection, moment, and shear of the structure when subjected to far-field dynamic loading.

Figure 4.1: Staad-Pro model

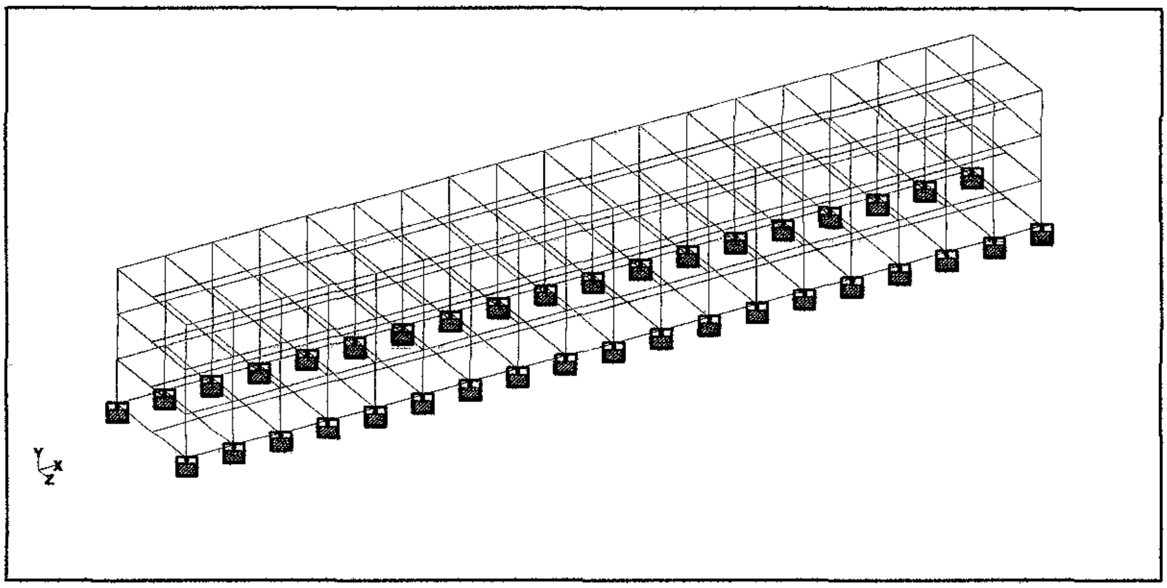
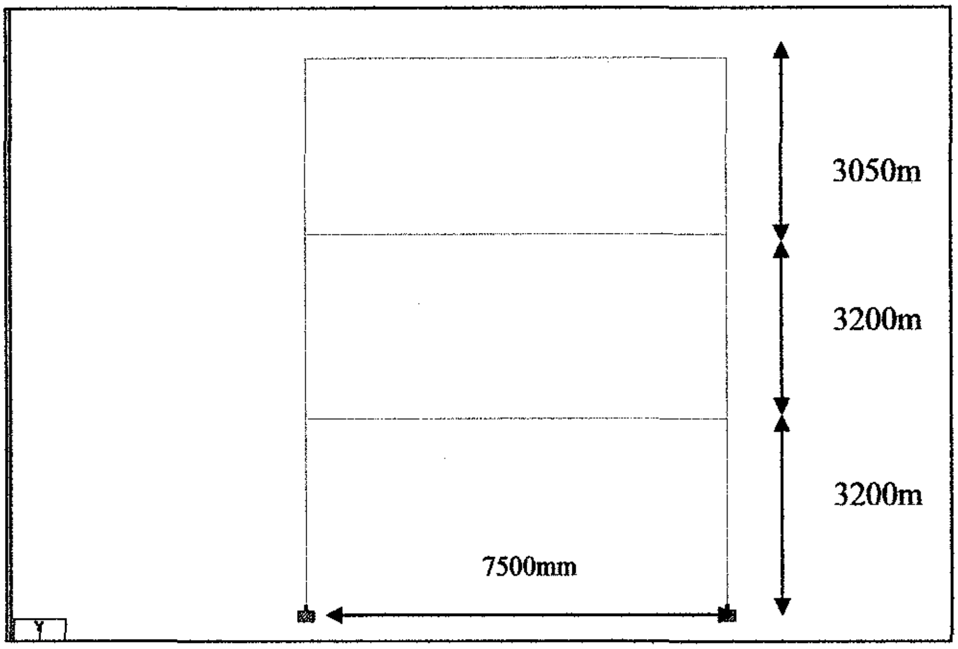


Figure 4.2 Side view of Frame 1 with Dead Load + Live Load + Seismic Load



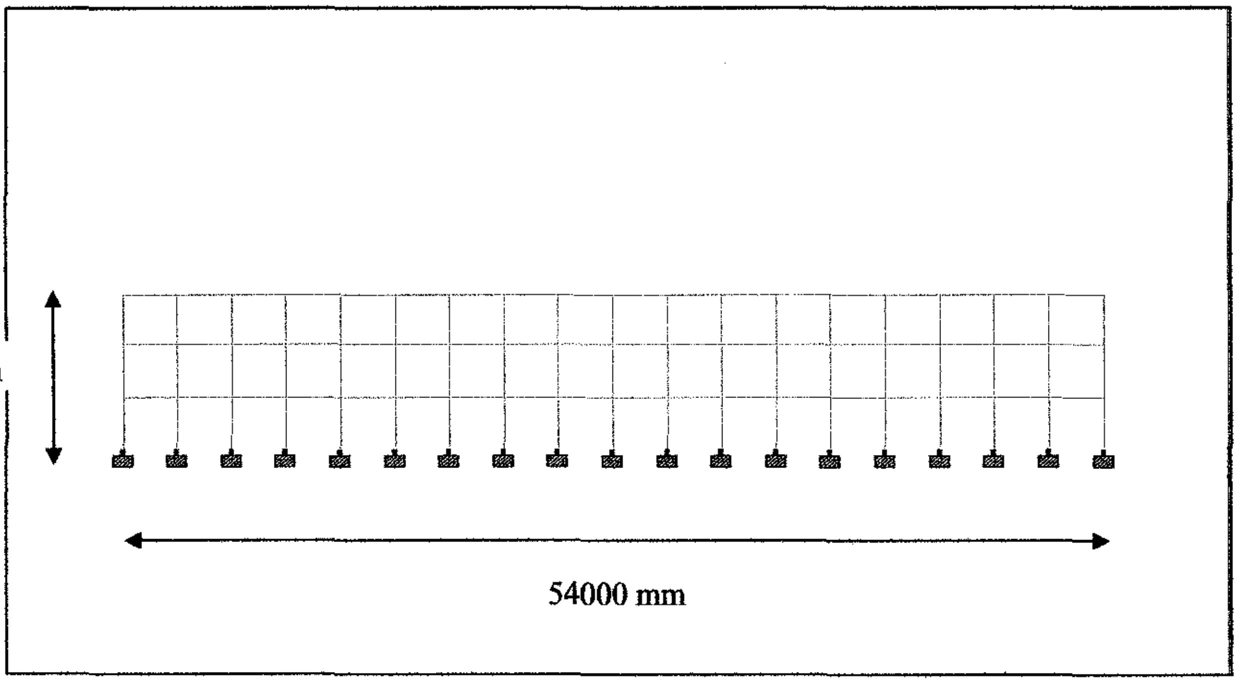
Member Sizes

Beams: 1^{st} floor and 2^{nd} floor = 200 mm X 550 mm

Roof = 200 mm X 400 mm

Columns: 250mm X 350 mm

Figure 4.3: Front View of Frame 1 with Dead Load + Live Load + Seismic Load



Member Sizes

First and Second Floor Beams: 550 mm X 200 mm

Roof Beam: 600 mm X 150 mm

Columns: 250mm X 350 mm

Loads Calculation

DEAD LOAD

i. 1st floor and 2nd floor beam

Total Dead Load = Beam Self weight + Load from slab + Brick wall + Finishes Dead Load

= (density of concrete x depth x width) + (slab self weight +load distributed by slab) + (density of brick x brick's thickness X wall height) + (Finishes)

= 20.284 kN/m

LIVE LOAD

As in BS 8110 code, for uniform school building (classroom) = 1.912 kN/m2

4.2 Deflection of column due to seismic load

Table 4.4 shows the result of the maximum deflection for 3 stories school building model in selected column. Column in structure are the most affected portion when subjected to lateral load. The result shows the deflection in X, Y and Z direction.

	Selected Column	Horizontal X	Vertical Y	Horizontal Z	Resultant (mm)
Max X	**200**	**29.437**	**0.001**	**0.000**	**29.437**
Min X	**172**	**-0.007**	**-0.790**	**-0.019**	**0.790**
Max Y	**172**	**29.429**	**0.293**	**0.002**	**29.431**
Min Y	**209**	**0.006**	**-11.506**	**0.000**	**11.506**
Max Z	**200**	**0.000**	**0.000**	**28.902**	**28.902**
Min Z	**210**	**-0.007**	**-0.790**	**-0.019**	**0.790**
Max Resultant	**200**	**29.437**	**0.001**	**0.000**	**29.437**

Table 4.4 Deflection of column due to seismic load

The maximum deflection is at column no 200 with the resultant of 29.437 mm. The maximum allowable deflections for the structural members due to Uniform Building Code (UBC) 1997 are as below:

$$\text{Max deflection} = \frac{\text{length of structure}}{240}$$

$$= \frac{54000}{240}\ \text{mm}$$

$$= 225\ \text{mm}$$

This means that the structures are still able to withstand the dynamic loading during the earthquakes. This building is still safe to be used.

Figure 4.5: Deflection about Z-axis

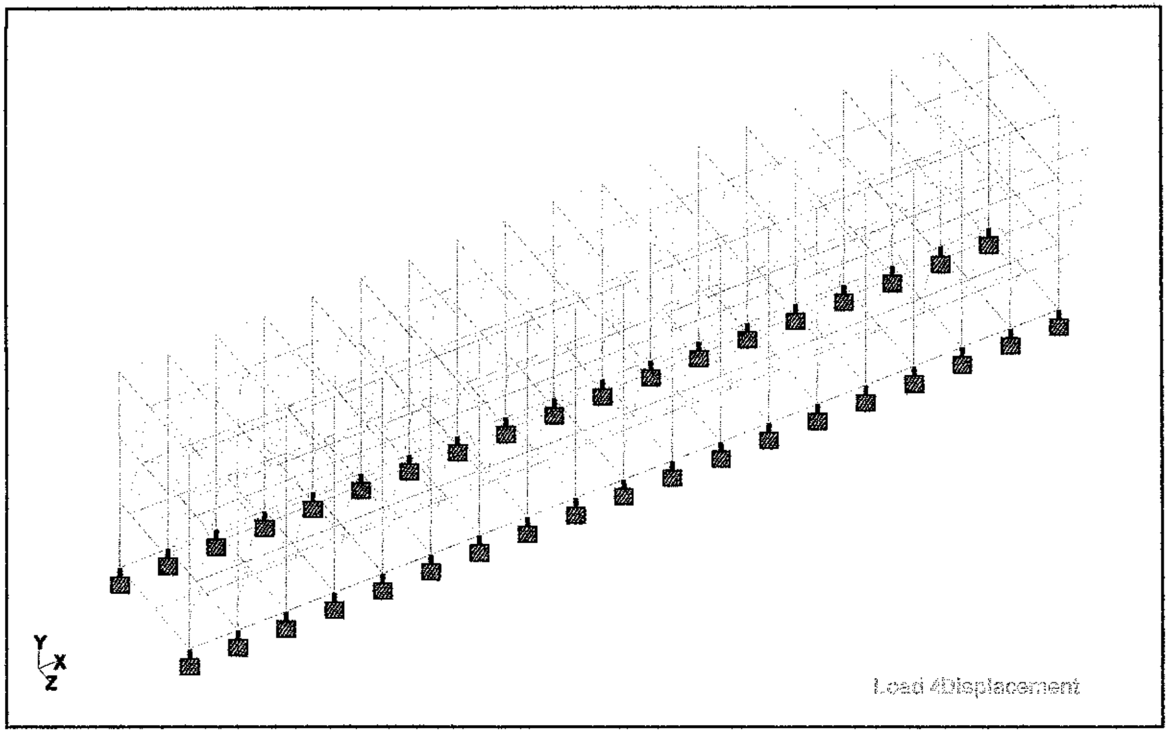
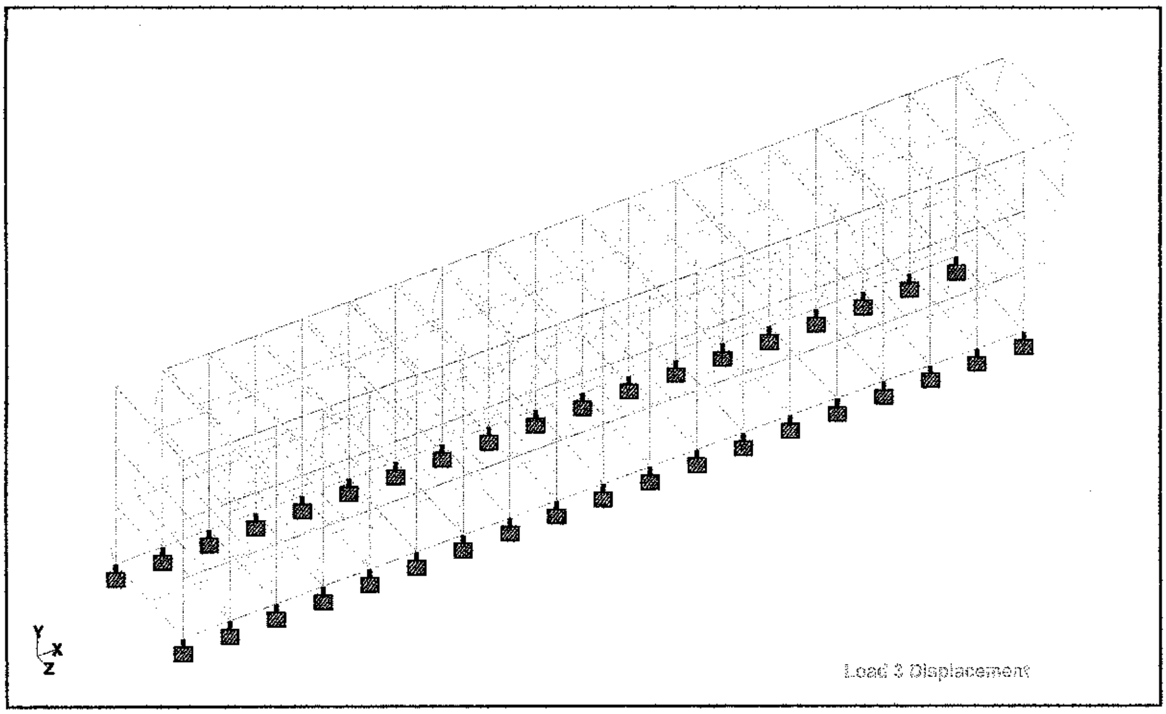


Figure 4.6: Deflection about X-axis



4.3 Bending Moment of column due to seismic load

The bending moment at a section through a structural element may be defined as the sum of the moments about that section of all external forces acting to one side of that section. The maximum bending moment of selected column is in table 4.7.

Selected Column	Moment Y kN.m	Column	Moment Z kN.m
416	109.618	370	77.435
415	109.532	408	77.435
414	109.509	424	77.201
417	109.443	386	77.201
419	109.364	408	77.201
413	109.316	370	77.201
420	109.308	386	76.966
412	109.125	424	76.966

Table 4.7 Bending moment of column due to seismic load

The maximum bending moment in structure was 109.618 kN.m. This maximum value occurred at the middle of the structure. Manual calculation for static capacity of the column shows that maximum bending moment at this specific column is about 150 kN.m. This means that the maximum bending moment occurred due to dynamic loading are still not exceeding the static capacity of the column.

4.4 Shear Force of column due to seismic load

The maximum shear forces for the selected column in the structure were shown in table 4.8. The table show shear force in decreasing order from maximum value until minimum value. The maximum shear force occurred at the edge of the structure. The shear forces of column decrease while it reaching the middle of the structure.

Column	Shear Y kN	Column	Shear Z kN
370	50.512	416	38.872
408	50.512	415	38.945
424	50.275	414	38.806
386	50.275	417	38.898
408	50.275	419	39.137
370	50.275	413	38.813
386	50.038	420	38.813
424	50.038	412	38.703

Table 4.8 Shear force of column due to seismic load

Maximum value for shear force due to seismic loading is 50. 512 kN. Since the value is still lower than maximum allowable static shear force from manual calculation, the building is still safe under far-field earthquake loading.

4.5 Axial Forces due to seismic load

The axial forces of selected column were shown in table 4.9 below. The value is decreasing from maximum value till the minimum value. The maximum value occurred at the middle column of the structure and decreasing to the edge of the structure.

Column	Axial Force, kN
379	621.864
417	621.864
424	618.608
386	618.607
408	618.607
370	616.117
410	616.117
372	616.110
422	616.110
384	615.470
373	615.470

Table 4.9 Axial force of column due to seismic load

4.6 Discussion

In the calculation for beam and column capacities, all the required information such as concrete cover thickness, grade of concrete, size of reinforcements are taken from the actual structural drawing of the school building. The results from the manual calculation are then compared with the value obtained in STAAD Pro analysis to determine if the current capacity is enough or safe to withstand seismic during earthquakes.

Before starting the STAAD Pro analysis, certain standard values of load such as live load for school building are to be found in ACI (2000) code, BS 8110 and 6399 for certain standard values. Manual Calculations are then performed to calculate the dead load imposed on the beam, which came from the beam self weight, finishes, as well as load from slabs. All of these values are then being input to STAAD Pro for analysis of the frame.

In the analysis, the seismic load is defined first before the execution of analysis. The seismic load is defined in terms of zone number, Rw in x-direction; y-direction and z-direction, soil factor and value of Ct. All of the values are referred to Uniform Building Code 1997.

In Post-Analysis Results, the bending moment, shear forces and axial forces enveloped on beams are obtained. When compared with the value of manual calculation, it was found the current capacity of beams and columns are still able to withstand the seismic load.

CHAPTER 5
CONCLUSION

A simple set of computer models has been developed for the analysis and design of reinforced concrete medium-rise building using STAADPro 2004 software. The model has been generated and the analysis has been performed with the software. The floor slabs are modeled to be rigid in plane, so that the columns do not have any rotation at the slab junctions.

From the analysis of the three-storey school building in Malaysia that have never been designed for earthquake resistant, the beams and columns are lightly affected by the far-field earthquake loading. The deflection, bending moment and shear force of the structure are not exceeding the maximum limit that has been design. The school structure is still able to withstand the loading and still safe to be used.

However, the accuracy of the result can be improved by providing values that are nearer to the exact situation in the real structure. Most of the dead loads such as roof load are standard values taken from BS 6399 and ACI Code. By incorporating the real value of materials used in the building, a more accurate value would be obtained in the calculation process, thus increasing the accuracy of the analysis under seismic load.

5.2 Future Recommendation

Further study is needed in order to investigate the effect of earthquakes to the higher building. Further works requires as listed below:

i. This study only on three storey building and further study on higher building need to be done.
ii. This study only considers earthquake load. It is necessary to make comparison between wind load and earthquake load in order to obtain more beneficial results.
iii. To change the building material such as steel and composite in further research.
iv. To analyze the building response due to seismic load in the form of time-history load.

REFERENCE

i. British Standard BS 6399 Part 1: 1996: *"Design Loading for Buildings, live load desired in analysis"*. BSI

ii. British Standard Institution, BS 8110: Part 1: 1997, *"Structural use of concrete, Part 1. Code of practice for design and construction"*. BSI

iii. Dianfeng Zhao and Kevin K.F Wong (2004). "New Approach for Seismic Nonlinear Analysis of Inelastic Framed Structures" *Journal of Engineering Mechanic*, Vol.132, No.9, September 2006, pp. 959-966

iv. Gustavo J. Parra-Montesinos, Sean W. Perterfreund and Shih Ho Chao (2004). "Highly Damage-Tolerant Beam-Column Joints through Use of High Performance Fiber Reinforced Cement Composites". *ACI Structural Journal*, V.102, No.3, May-June 2005, pp.487-495

v. Jack C. McCormac and James K. Nelson (2005). *Design of Reinforced Concrete*. Sixth Edition. John Wiley & Sons, Inc

vi. Laura N. Lowes and Aras Altoontash (2003). "Modeling Reinforced Concrete Beam-Column Joints Subjected to Cyclic Loading". *Journal of Structural Engineering*, December 2003, pp. 1686- 1696

vii. Megawati, K., Pan, T.C & Koketsu, K. (2005). Response Spectral Attenuation Relationship for Sumatran-subduction Earthquake and The Seismic Hazard Implications to Singapore and Kuala Lumpur. Soil Dynamics and Earthquake Engineering. 25, pp 11-25.

viii. National Earthquake Hazard Reduction Program, NEHRP (1997)

ix. Peterson, M. D., James Dewey, Hartzell, S., Mueller, C., Harmsen, S., Frankel, A.D, Rukstales, K. (2004). Probabilistic Seismic Hazard Analysis for Sumatra, Indonesia and Across the Southern Malaysia Peninsula. Tectonophysics. 390, pp 141-158.

x. Rashwan et al, 2007. Development of Design Response Spectra for Various Soil Type in Ipoh, AWAM '07

xi. Sekolah Piawai (3-tingkat) Ibu Pejabat JKR Malaysia, Cawangan Kerja Pendidikan.

xii. Smith, B. S and Coull, A., 1991. Tall Building Structures: Analysis and Design. John

xiii. Taranath B.S., 2005. Wind and Earthquake resistant buildings, structural analysis and design. Marcel Dekker. pp 99 – 390

xiv. Uniform Building Code Earthquake regulations for seismic isolated structures. Whitter (CA); 1997

xv. Yan Xiao and Armen Martirossyan (1998). "Seismic Performance of High-Strength Concrete Column". *Journal of Structural Engineering/ March 1998* (pp.241-250)

xvi. Yong Lu (2002). "Comparative Study of Seismic Behavior of Multistory Reinforced Concrete Framed Structures". *Journal of Structural Engineering,* February 2002, pp. 169-178

APPENDIX A

FORMULAS FOR

MANUAL CALCULATIONS

Formulas

Formulas involved in calculation of beam and column capacities:

1. Beams

Ultimate Moment Strength, Mn

- $M_n = T\,(d - a/2)$ (unit: kN.m)
  - Tension, $T = A_s . f_y$
  - Compression, $C = 0.85 f_c . a . b$
  - a is obtained from equation when T=C (equilibrium)

- T- Beams

  $\varnothing M_n = \varnothing T z$ (unit: kN.m)
  - Tension, $T = A_s f_y$
  - Area of concrete in compression, $A_c = T/0.8 f_c$
  - $a = Ac/b_w$; b_w is the effective width
  - $z = d - a/2$
  - Reduction factor, $\varnothing = 0.90$

2. Columns

Ultimate Moment Strength, Mn

- $As = 4 \times \Pi \times (20)$

  $= 1257$ mm2

- $\frac{As}{Bd} = \frac{1257 \times 100}{350 \times 250}$

  $= 1.43\ \%$

- N = Load = 400kN

- $\frac{N}{Bd} = \frac{400 \times 1000}{350 \times 250} = 4.57$

- $\frac{M}{Bh^2} = 5$

  $M = \frac{5 \times 250 \times 350^2}{1 \times 10^6}$

  $= 153$ kN.m

Shear, V_u

- $Vu = ØV_c + ØV_s$ (unit: kN)
  - $Vc = (\sqrt{fc}/6)\, b_w d$ (In SI)
  - $Vs = ⅓ \sqrt{fc}\, b_w d$ (In SI)
  - $Av = \frac{Ø \sqrt{fc}}{12} \frac{A^2 cp}{p_{cp}}$
  - $Ø = 0.75$

APPENDIX B

INSTRUCTION IN

CONSTRUCTING AND ANALYSIS

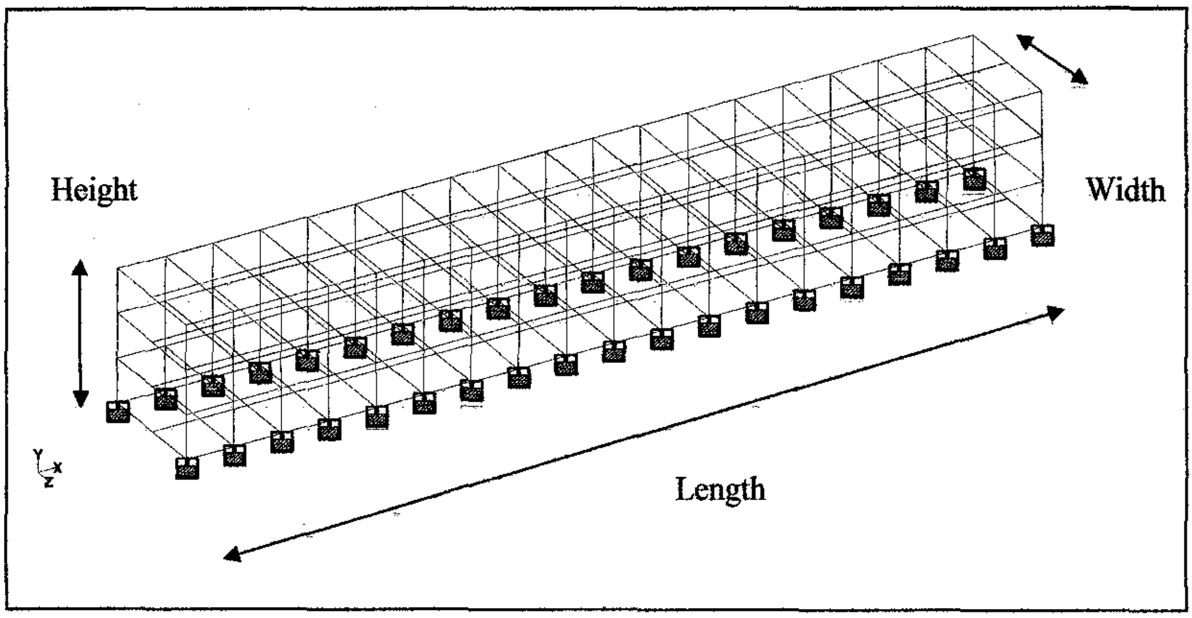
OF MODEL USING

STAADPRO 2004

1.0 Creating Model Using Graphic

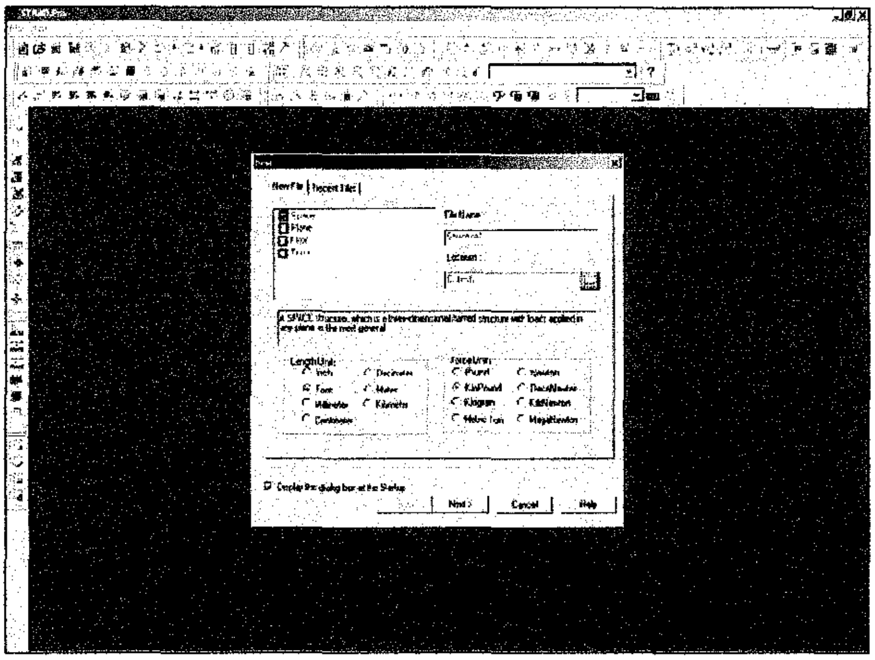
Example of Model

1. **Constructing Model**



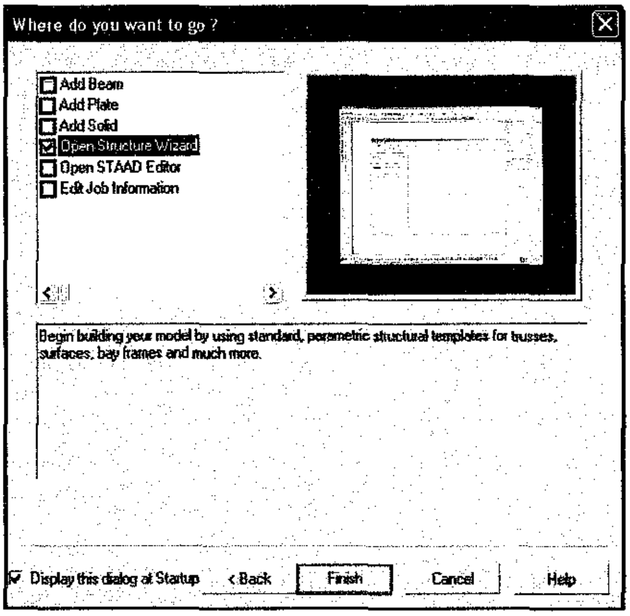
* **Step :**

1. New



2. Space (3D)

3. Open Structure Wizard



- choose model type – frame models – grid frame

Length: 54 m	No. of bays along length: 18
Height: 9 m	No. of bays along height: 3
Width: 7.5 m	No. of bays along width: 1

File – Merge model with STAAD.Pro model

Yes – to confirm

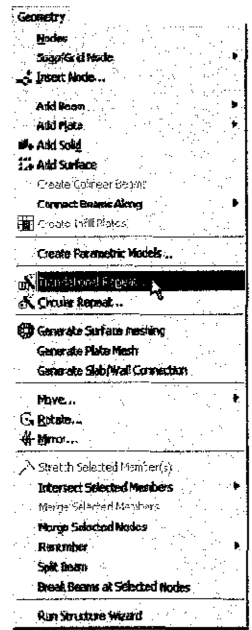
4. Slab

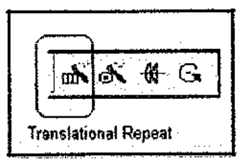
  - Cut section – X-Z plane – with node 1 (to show the lowest level)
  - Add 4-noded plates and 3-noded plates – apply into the node (apply into the same direction)

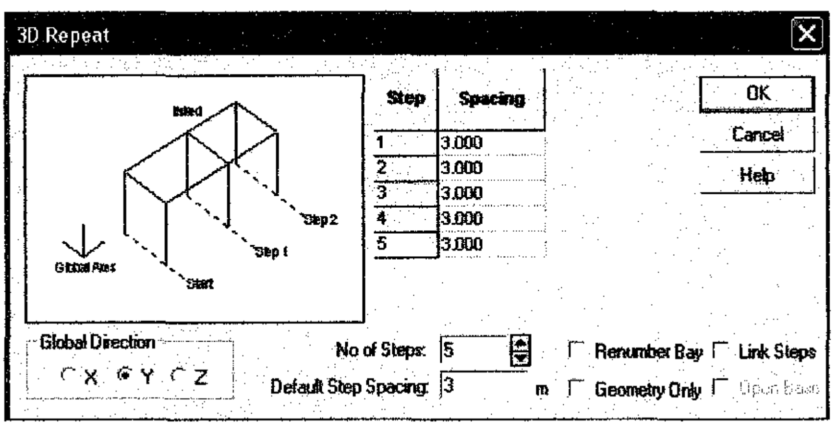
- Copy plate at each levels :
  - Select plates using plate cursors
  - Translational repeat – global direction = Y

    - No of steps = 5

    - Default spacing = 3 m

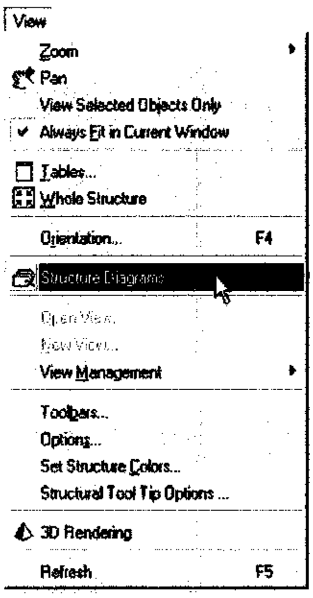

Geometry
Nodes
Snap/Grid Node
Insert Node...
Add Beam
Add Plate
Add Solid
Add Surface
Create Collinear Beams
Connect Beams Along
Create Infill Plates
Create Parametric Models...
Translational Repeat...
Circular Repeat...
Generate Surface meshing
Generate Plate Mesh
Generate Slab/Wall Connection
Move...
Rotate...
Mirror...
Stretch Selected Member(s)
Intersect Selected Members
Merge Selected Members
Merge Selected Nodes
Renumber
Split Beam
Break Beams at Selected Nodes
Run Structure Wizard

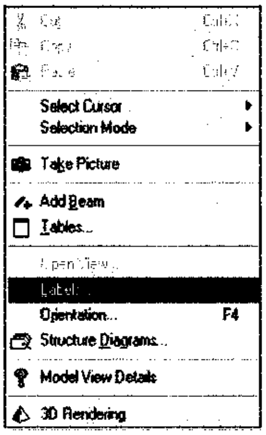

Translational Repeat


3D Repeat
Step
Spacing
1 3.000
2 3.000
3 3.000
4 3.000
5 3.000
OK
Cancel
Help
Step 2
Step 1
Start
Global Axes
Global Direction
X Y Z
No of Steps: 5
Default Step Spacing: 3 m
Renumber Bay
Link Steps
Geometry Only

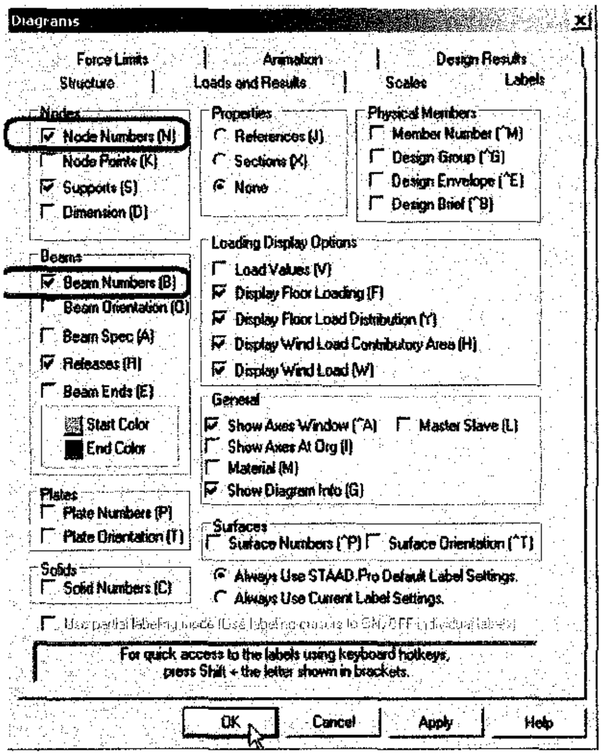
2.0 Switching on node and beam labels

- Node and beam labels are a way of identifying the entities we have drawn on the screen. In order to display the *node* and *beam numbers*, right click anywhere in the drawing area. In the pop-up menu that comes up, choose **Labels**. Alternatively, one may access this option by selecting the *View* menu followed by the *Structure Diagrams* option from the top menu bar, and the *Labels* tab of the dialog box that comes up.





1. In the *Diagrams* dialog box that appears, turn the **Node Numbers** and **Beam Numbers** on and then click on **OK**.



3.0 Specifying member properties

Steps:

1. To define member properties, click on the **Property Page** icon located on the top toolbar.

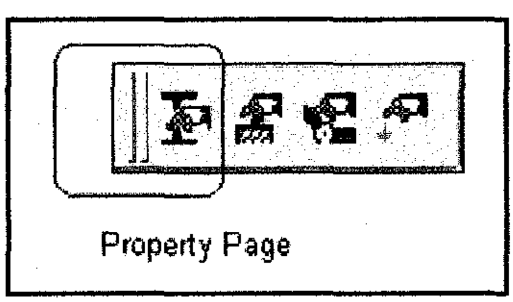
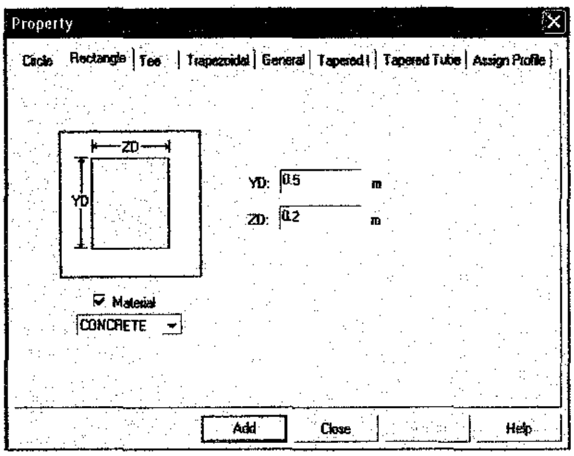
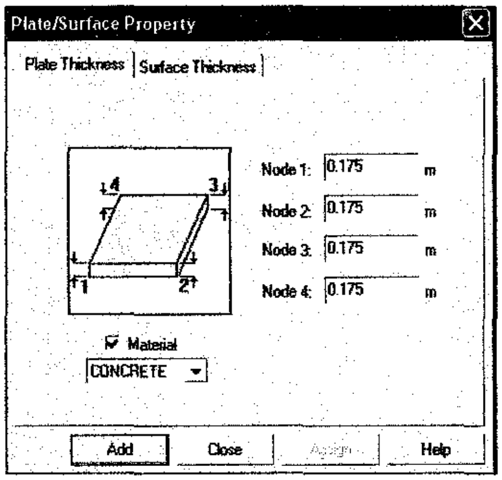


Figure 1. 19

- Column and beam
  - define – rectangular – (size YD 0.5 m, ZD 0.2 m)



  - add – close
  - select - beam parallel to Y/X/Z (which is included)
  - assign to selected beams
- slabs and lift core / shear wall



  - thickness – (0.175 m)
  - add – close
  - select plate in window
  - assign to selected plate

4.0 Specifying material constants

At the time of assigning member properties, we deliberately chose not to assign the material constants simultaneously, since we wanted to specify values which are different from the built-in defaults. The desired values are listed at the beginning of this tutorial. The corresponding commands we wish to generate in the STAAD input file are:

CONSTANTS

E 22 ALL

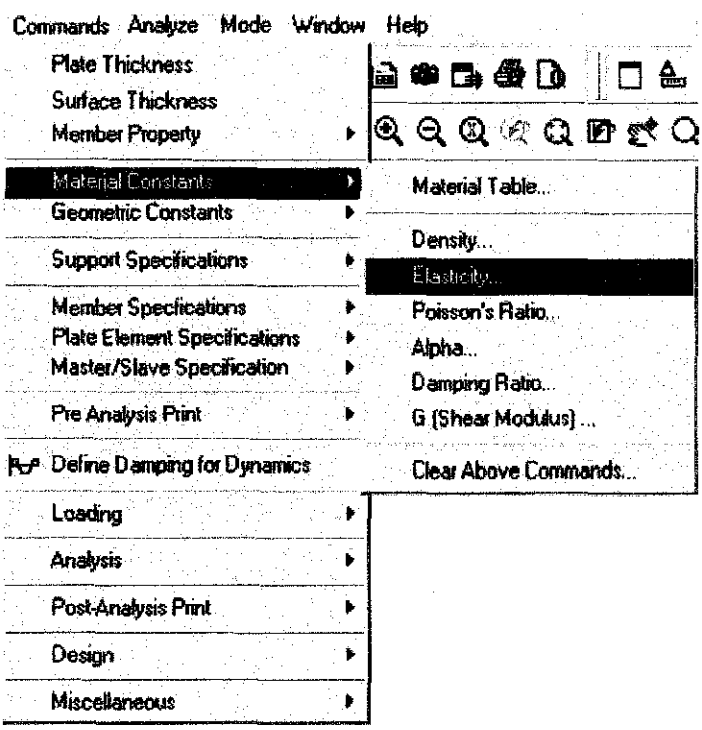
UNIT METER

DENSITY 25.0 ALL

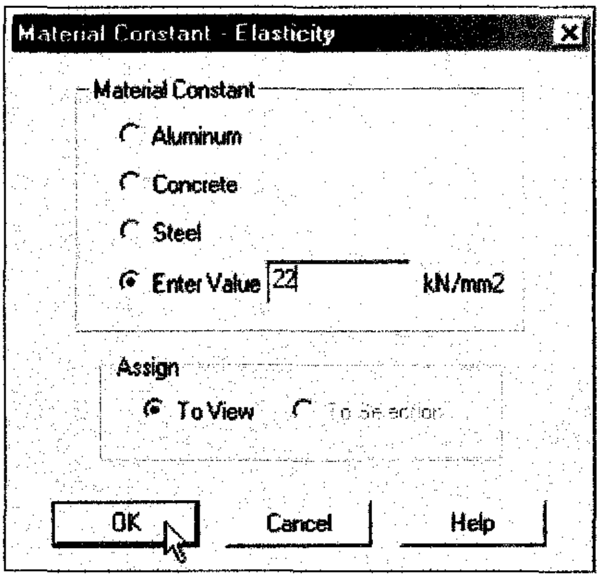
POISSON 0.17 ALL

Steps:

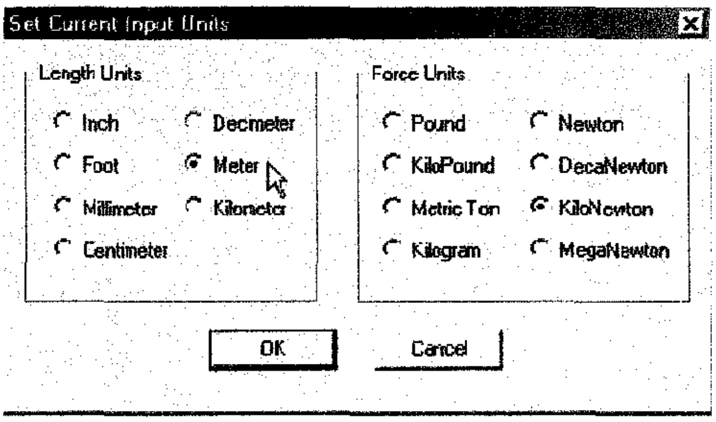
1. From the **Commands** menu, select **Material Constants.** To define the Modulus of Elasticity, select the **Elasticity** option as shown below.



2. In the *Material Constant* dialog box that appears, enter **22** in the *Enter Value* box. Since the value has to be assigned to all the members of the structure, the current setting of the assignment method, namely, **To View**, allows us to achieve this easily. Then, click on **OK**.



3. For specifying the DENSITY constant, it will be convenient if we change our length units to meters. To change the length units, as before, click on the **Input Units** icon from the Structure toolbar, or select the *Tools* | *Set Current Input Unit* menu option from the top menu bar. In the *Set Current Input Units* dialog box that comes up, specify the length units as **Meter**.

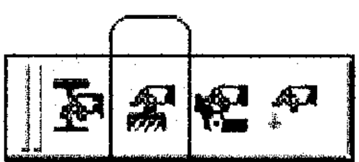


4. Following the steps 1 and 2 above, we choose **Commands |Material Constants | Density**, specify the value as **25KN/m3**, and assign **To View**.

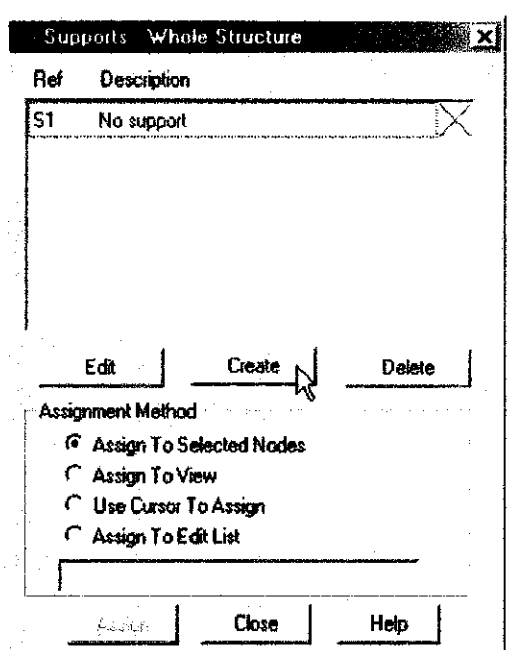
5. To define the **POISSON'S RATIO**, using the similar procedure as described above, provide the value **0.17** to all members in the **View**.

5.0 Specifying Supports

- support page – create – fixed – add – use cursor to assign – assign to the node



Support Page

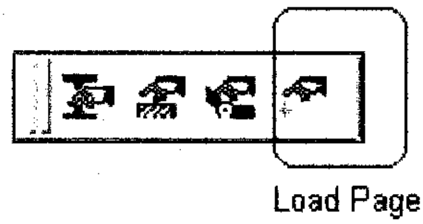


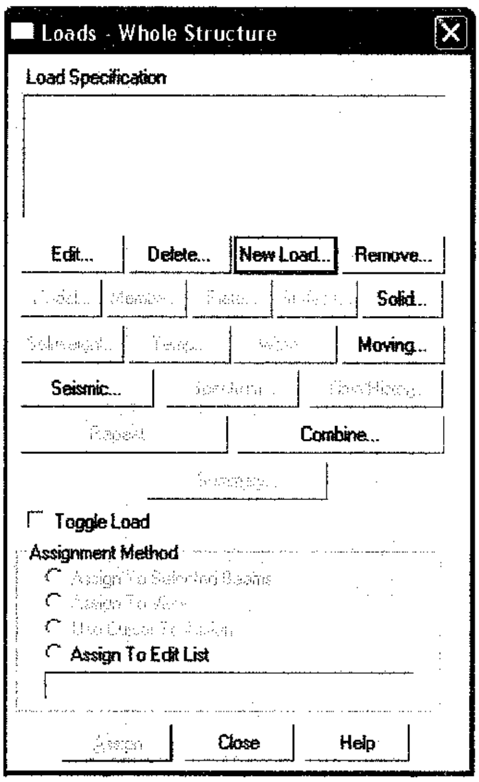
6.0 Specifying Loads

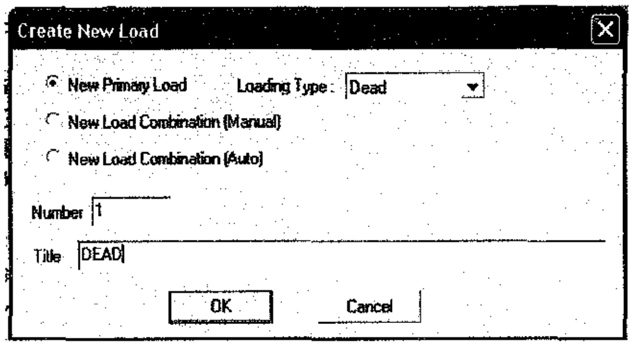
Steps:

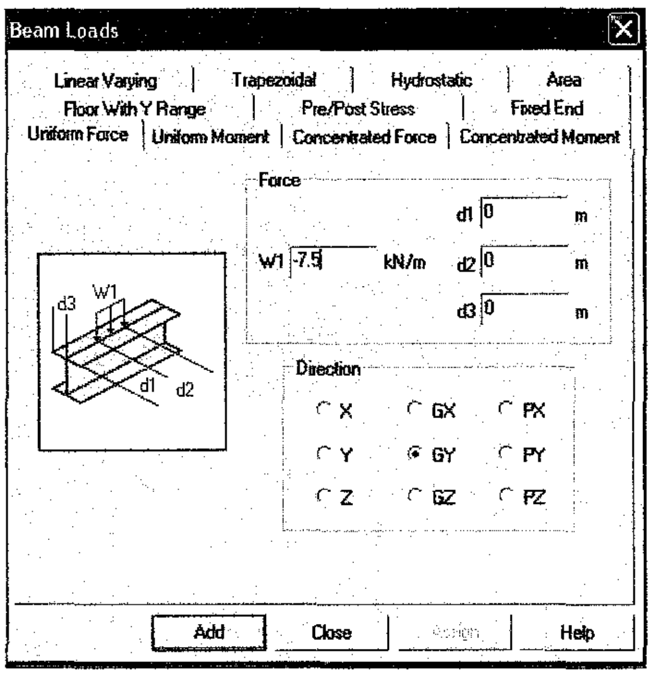
LOAD CASE 1

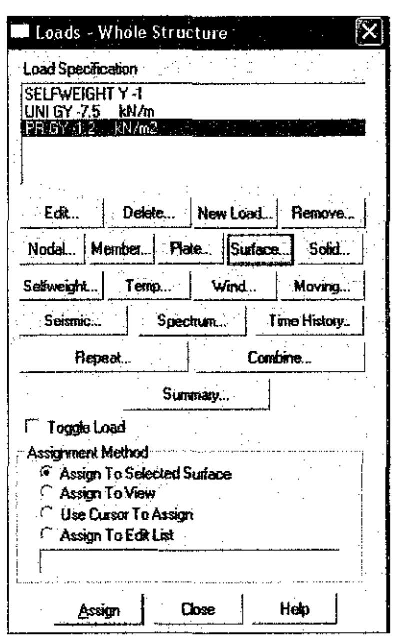
1. To create loads, click on the **Load Page** icon located on the *Structure Tools* tool bar.





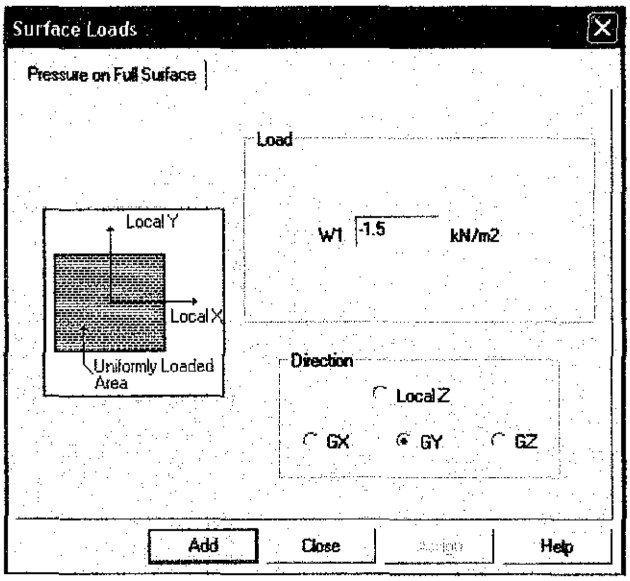

Create New Load
New Primary Load
Loading Type : Dead
New Load Combination (Manual)
New Load Combination (Auto)
Number 1
Title DEAD
OK
Cancel

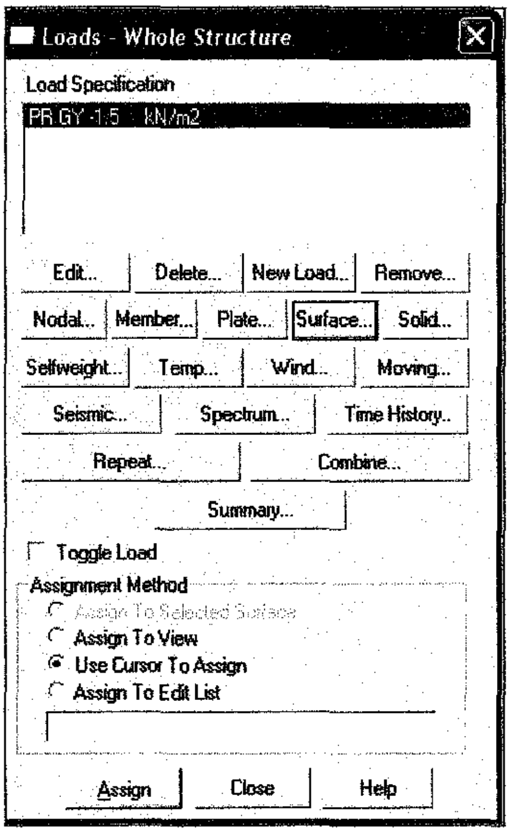

Beam Loads
Linear Varying
Trapezoidal
Hydrostatic
Area
Floor With Y Range
Pre/Post Stress
Fixed End
Uniform Force
Uniform Moment
Concentrated Force
Concentrated Moment
Force
d1 0 m
W1 -7.5 kN/m
d2 0 m
d3 0 m
d3
W1
d1
d2
Direction
X
GX
PX
Y
GY
PY
Z
GZ
PZ
Add
Close
Assign
Help


Loads - Whole Structure
Load Specification
SELFWEIGHT Y -1
UNI GY -7.5 kN/m
PR GY -1.2 kN/m2
Edit...
Delete...
New Load...
Remove...
Nodal...
Member...
Plate...
Surface...
Solid...
Selfweight...
Temp...
Wind...
Moving...
Seismic...
Spectrum...
Time History...
Repeat...
Combine...
Summary...
Toggle Load
Assignment Method
Assign To Selected Surface
Assign To View
Use Cursor To Assign
Assign To Edit List
Assign
Close
Help

LOAD CASE 2 : LIVE LOADS

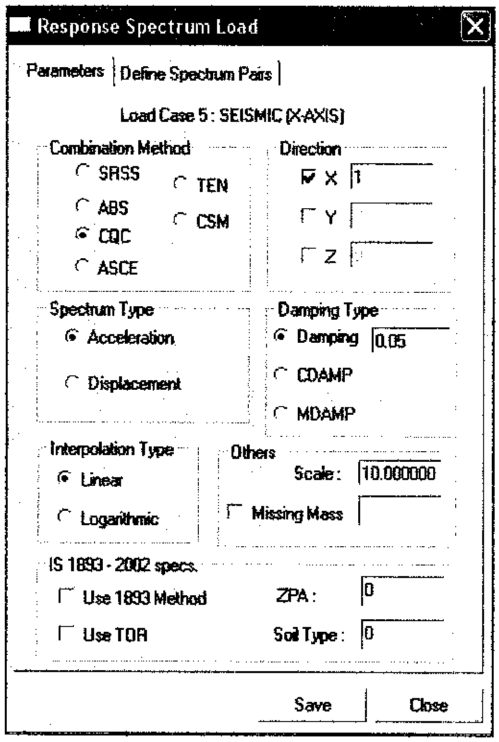
- New load – live load - surface - -1.5 - GY

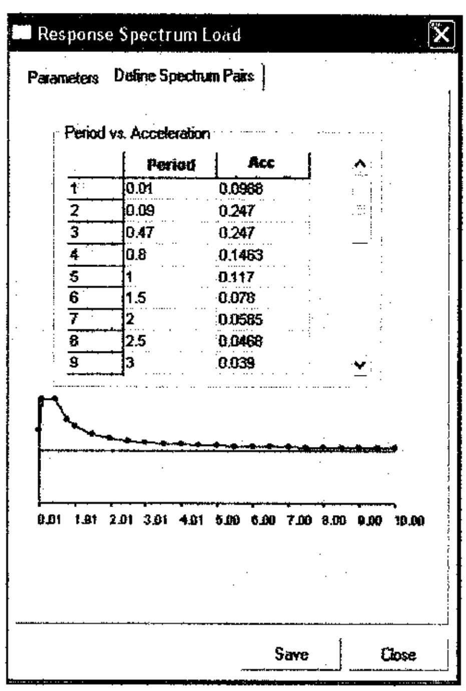


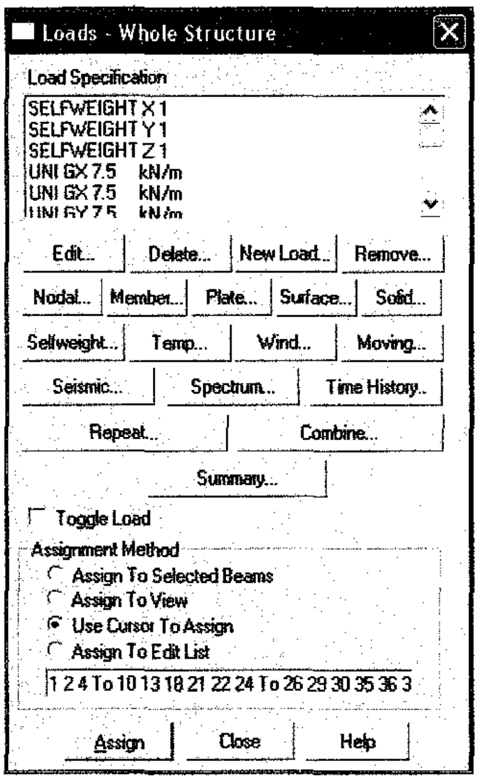


LOAD CASE 3 : SEISMIC LOAD (RESPONSE SPECRUM)

1. Seismic Load = Dead load + response spectrum
2. Apply dead load a in load case 1 (but in positive direction)
3. Apply response spectrum:

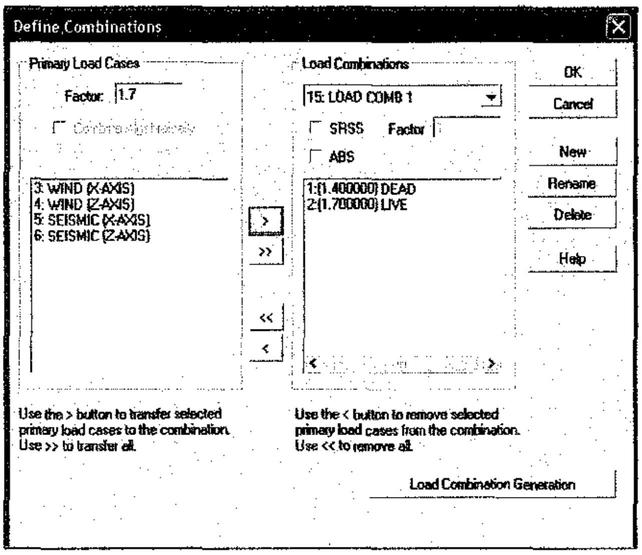






LOAD COMBINATION 1

- From loads windows – combine



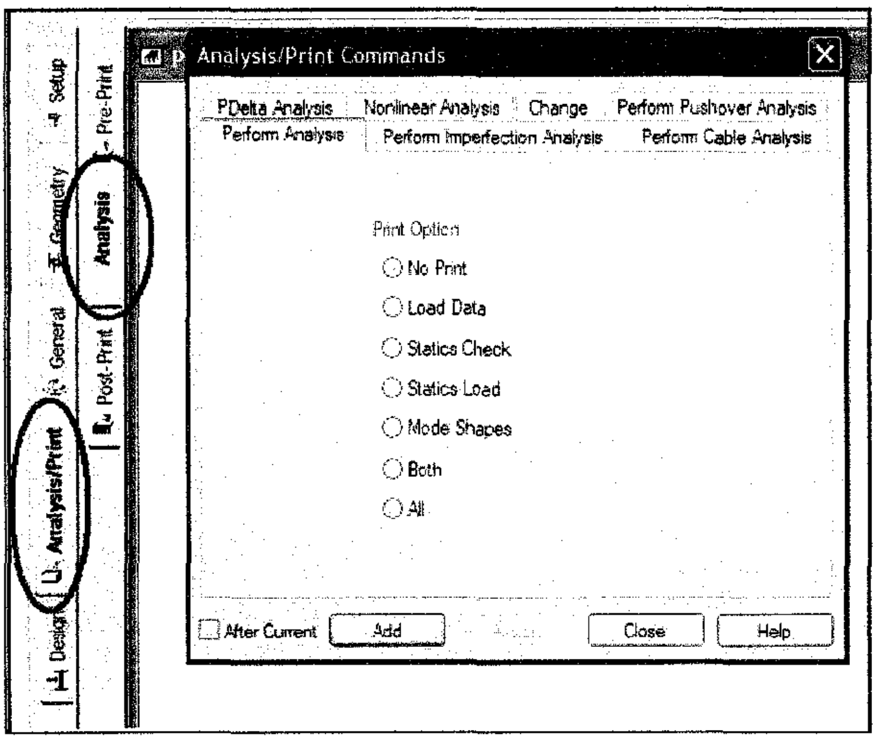
7.0 Specifying the analysis type

- The analysis type we are required to do is a linear static type. We also need to obtain a static equilibrium report. This requires the command:

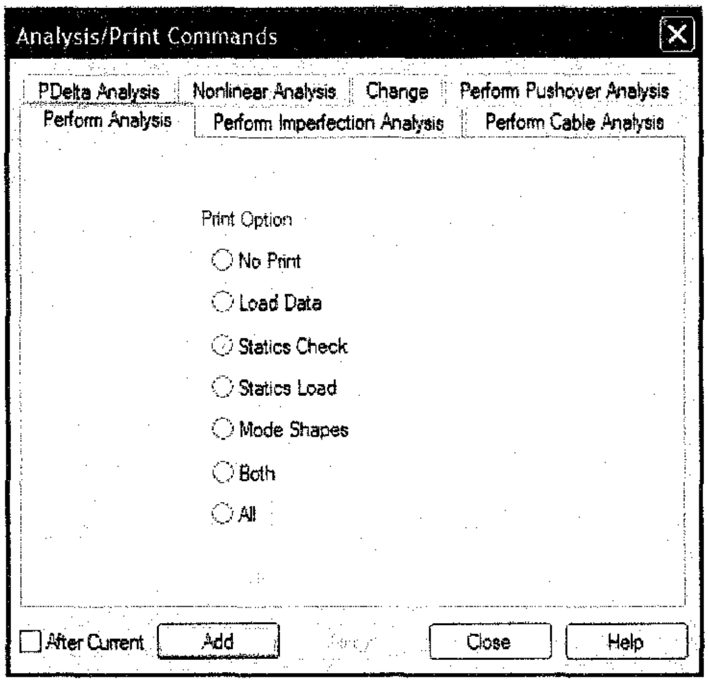
PERFORM ANALYSIS PRINT STATICS CHECK

Steps:

1. To specify the Analysis command, go to **Analysis/Print** Page from the left side of the screen. By default, the **Analysis** sub-page from the second row is in focus as shown below.



2. In the *Analysis/Print Commands* dialog box that appears, make sure that the **Perform Analysis** tab is selected. Then, check the **Static Check** print option. Finally, click on the **Add** button followed by the **Close** button.

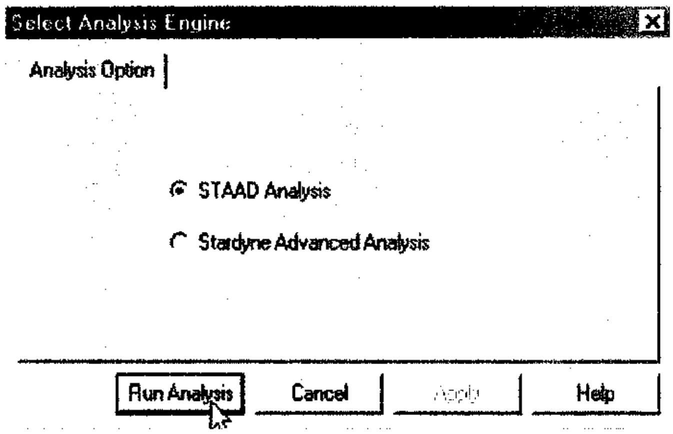


Let us save the data once again using the **File | Save** option.

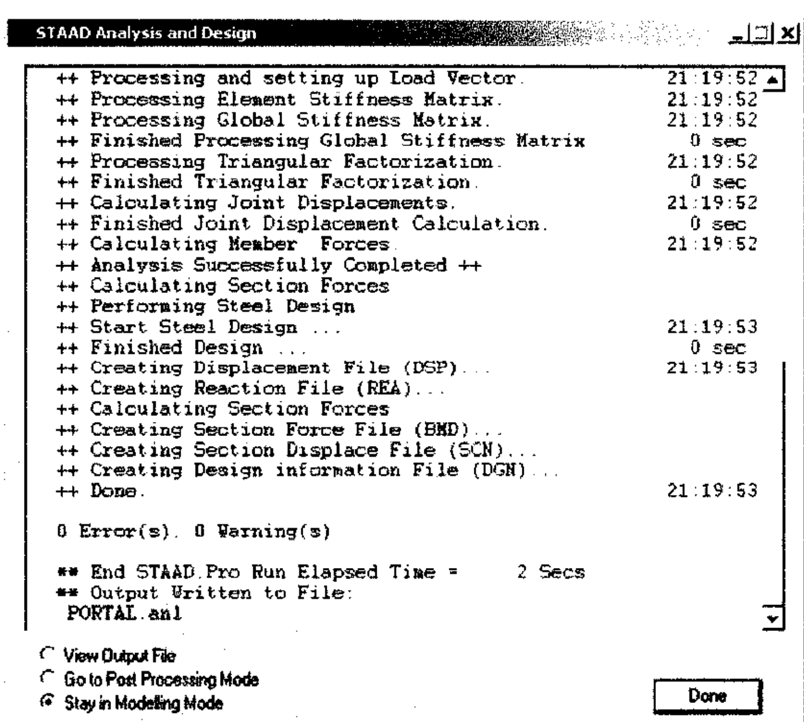
8.0 Performing Analysis/Design

- STAAD.Pro performs Analysis and Design simultaneously. In order to perform Analysis and Design, select the **Run Analysis** option from the **Analyze** menu.

- If the structure has not been saved after the last change was made, you should save the structure first by using the *Save* command from the *File* menu.When you select the *Run Analysis* option from the *Analyze* menu, the following dialog box appears:

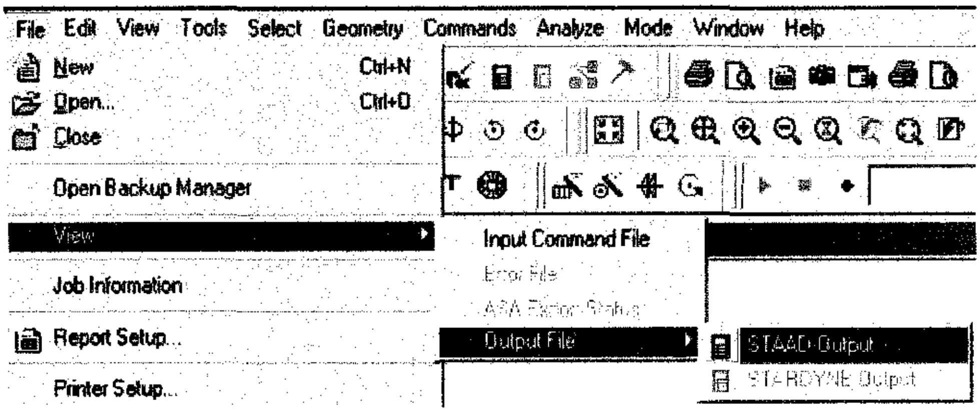


- We are presented with the choice of 2 engines : the STAAD engine and the STARDYNE Advanced Analysis engine. The STARDYNE Analysis engine is suitable for advanced problems such as Buckling Analysis, Modal Extraction using various methods, etc. However, if the calculations call for steel or concrete design, UBC load generation, etc., we have to select the STAAD engine. So, let us ensure that the radio button is on the **STAAD** engine.
- Click on the **Run Analysis** button. As the analysis progresses, several messages appear on the screen as shown in the figure below.



9.0 Viewing the output file

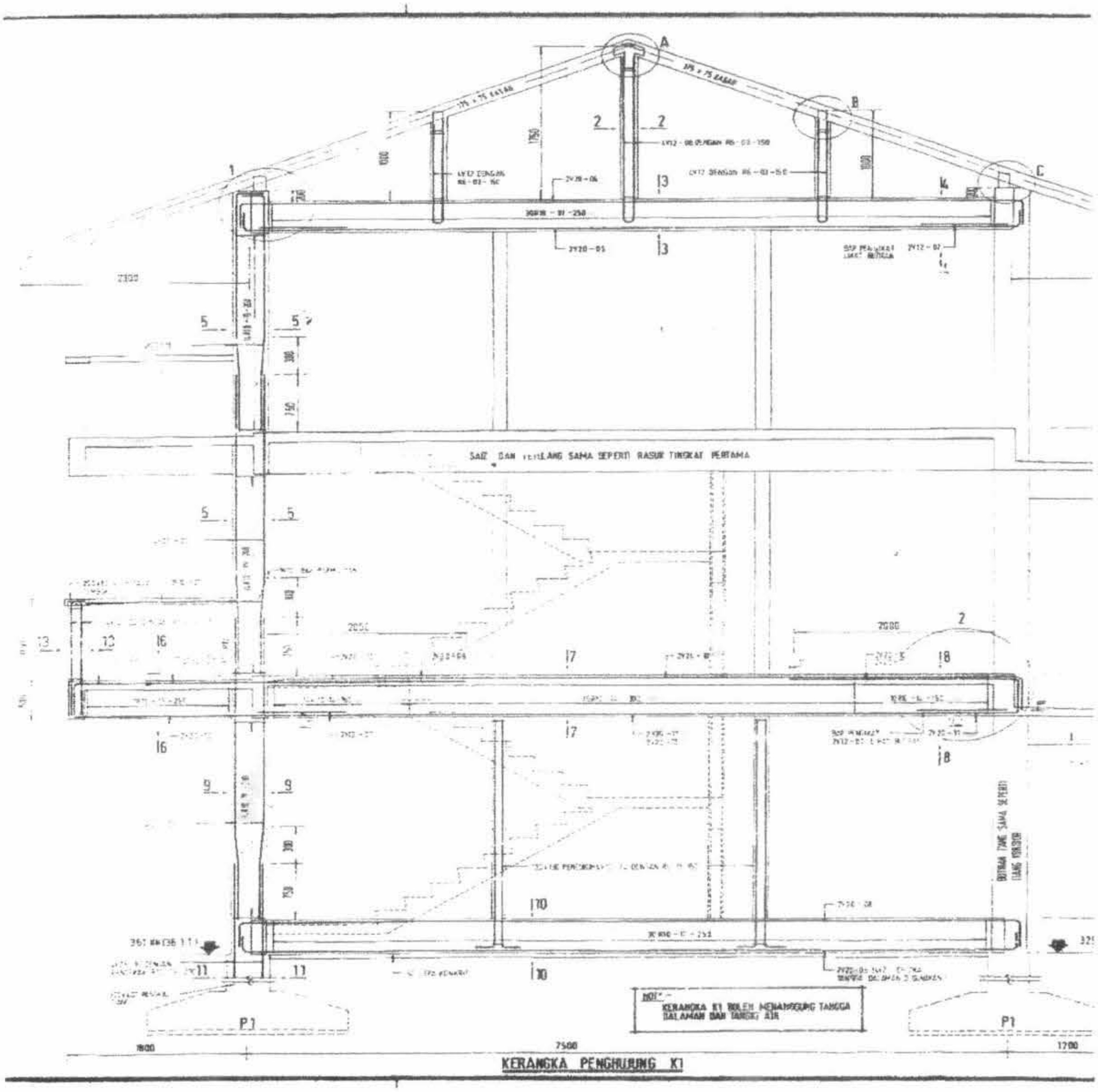
During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis was performed properly. For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file. We can access the output file using the method explained at the end of the previous section. Alternatively, we can select the **File | View | Output File | STAAD Output** option from the top menu. The STAAD.Pro output file for the problem we just ran is shown in the next few pages.



APPENDIX C

STANDARD SCHOOL BUILDING


F


KERANGKA PENGHUJUNG K1
P1
P1
7500