Seismic Analysis of Typical Steel Frame Building

Rabie A. Amnisi*, Jalal Solman, Ibtisam.F.Abou-Ajaila

Department, Of Civil Engineering, University Of Derna

rabie@istc.edu.ly

الملخص

التحليل التفصيلي في هذة الورقة هو تحليل ثابت وتحليل شكلي وتحليل وقتي في برنامج ANSYS لمبني هيكلي معدني ودراسة استجابتة لتاثيرالزلزال وأيضًا تم التحقق من نتائج التحليل بواسطة الحسابات اليدوية ومقارنتها بنتائج برنامج التحليل ANSYS بدقة عالية بالاضافة الي اجراء تحليل نقدي مفصل خاصة في نمذجة الهيكل من خلال ANSYS و كذلك تم عرض النتائج بوضوح ومناقشتها بشكل نقدي في هذة الورقة تم إجراء تحليل تفصيلي وتم فحص البيانات مثل ترددات المبنى والكتلة المشاركة مع إجراء الحسابات اليدوية للتحقق من الفحوصات وردود الافعال للهيكل المعدني وتفاعلة مع حركة الزلزال بما في ذلك تأثيرات التخميد من Rayleigh (ولكن تم تجاهل تفاعل التربة والمبني) مع العلم ان تحليل التاريخ الزمني لتسريع القاعدة الثابتة للتكامل المباشر على الهيكل باستخدام وقت زمني للإزاحة محددة لمدة 10 ثوانٍ. أخيرًا تم استخدام فحص "الكود" باستخدام Euro Code لفحص ما إذا كان الهيكل يمكنه تحمل الحدث الزلزالي المحدد.

الكلمات المفتاحية: الهيكل المعدني، التحليل الثابت، التحليل الديناميكي، التأثير الزلزالي.

Abstract

The detailed analysis comprises of static analysis, modal analysis and transient analysis in ANSYS and also to verify the results of the analysis by hand calculations A critical analysis of how this procedure has been carried out especially in the modeling of the structure through ANSYS, and how the results have been obtained will be presented clearly. A modal analysis was performed and data such as the frequencies of the building and the participating mass was checked. Hand calculations verifying the checks above were also is performed including Rayleigh damping effects (but ignoring soil-structure interaction). A direct integration fixed-base acceleration time-history analysis on the structure using prescribed 10 second displacement time-history supplied. Finally, a 'code' check using Euro Code will be used to examine whether the structure can withstand the prescribed seismic event and that the column behave adequately..

Keywords: Damping Frequencies, Static analysis, Seismic analysis Nuclear

1. Introduction

To carry out seismic analysis of nuclear power station building shown in figure 1

In this paper a seismic analysis and assessment of typical steel frame building housing a number of vital generators for a Nuclear Power Station as shown in Figure (1) below.

The construction details for the structure are stated below:

All of the beams at the first floor are constructed from 254×146×43 UB^' s, and all of the beams at the top floor (attic) are 152×89×16UB^' s.

All columns are constructed from 254×254×107 UC^'s. The roof is constructed as a pitched

roof as shown in Figure 2 with profiled steel sheeting as shown Figure 3, supported on light gauge steel purlins (C Section) spaced at 1200mm centers on the roof slope (Assume 170mm deep purlins, 1.8mm thick, 275N/mm^2 steel).
The first floor is constructed of continuous concrete and can be assumed to be 200mm thick (ignoring openings for stairs etc.).
The top floor (attic) is a normal timber joist floor, and can be assumed to be decked with 12mm plywood.
The frame is designed as a rigid moment resisting frame, therefore no internal partitions contribute to bracing.
The floor finishes and internal partitions contribute a characteristic dead load of1kN/m^2, and assume an additional characteristic floor live load of 3kN/m^2 at the first floor. On the top floor assume 1kN/m^2 live load only.
External walls are formed from a special lightweight modular cladding which weighs 0.5kN/m^2. It can be assumed that this offers no structural bracing or load-bearing capacity; therefore it need only be included as an additional dead load on the beams.
All column bases are fixed
Density of concrete used is 2400kg/m^3
Density of steel used is 7800kg/m^3

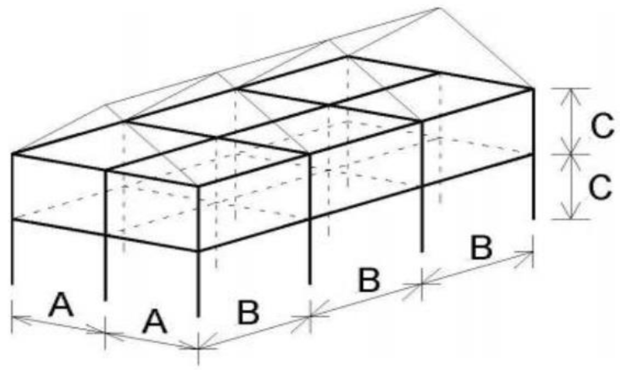


Figure.1. Steel Framed Building A=5.5m B=4m C=3m

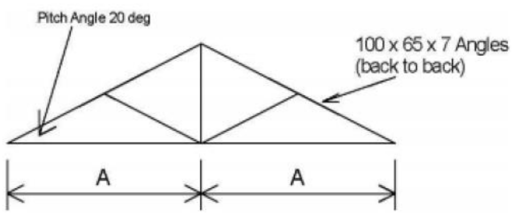


Figure 2. General arrangement of roof truss

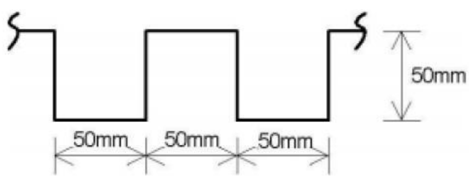


Figure 3. Approximate Profile of Roof Sheeting

Taucer et al [1991] proposed a fiber modeling approach where, the structural element is divided into a number of segments. The behavior of each segment is monitored at its centre cross section, which is divided into a number of fibers. A material model that accounts for yielding and strain hardening of steel is assigned for each fiber. The cross section response is determined by integrating the fiber responses over the cross section. Similarly, the element response is obtained by integrating the cross section responses along the element length. The fiber models are capable of providing accurate predictions of the element inelastic response. However, the only limitation associated with it is the substantial amount of computations required for monitoring the responses of several cross sections along the element length and the responses of several fibbers over each cross section.

2. Constructing a mathematical model of the building using ANSYS

The elements used for each of the structural elements are stated below with their various attributes.

2.1.1 BEAM 188

Beam 188 is suitable for analysing slender to moderately stubby/thick beam structures. Beam 188 is a linear (2-nide) or a quadratic beam in 3-D. It has six or seven degree of freedom at each node; these include translations in the x, y and z directions. This element is well suited for linear, large rotation, and/ or large strain nonlinear applications. Beam 188 includes stress stiffness terms, by default, the provided stress stiffness terms enable the elements to analyse flexural, lateral and torsional stability problems

2.1.2 SHELL 63

Shell 63 has both bending and membrane capabilities. Both in-plane and normal loads are permitted. The element has six degree of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. Stress stiffening and large deflection capabilities are included

TABLE1. Element type for each structural element

Beam 188	Shell
Beams (steel) Columns (steel) Timber joist Purlins (steel) Truss (steel)	Concrete Slab Plywood Floor Roofing Sheet

3. Material Properties

The materials used included steel, timber and concrete. The material attributes are tabulated in the table 2. However, the weight of the claddings used is converted to density and added to the density of the external steel beam. While creating the material attributes in ANSYS a different material is made for the external beams on the first floor as they only carry the cladding load.

TABLE2. Material properties

Material	Young's Modulus Nm^{-2}	Poisson Ratio (ν)	Density (kgm^{-3})
Concrete	2.6×10^{10}	0.2	2400
Steel	2.1×10^{11}	0.3	7800
Timber	1×10^{10}	0.12	720
External steel Beam	2.1×10^{11}	0.3	35970

4. Real Constant

The real constants were basically used to specify the properties of the shell element. The main fields entered specified the thickness and the additional weight it supports. These weights were either live loads or dead loads.

Pre-processor > real constants > add edit delete the table below shows an overall description of the elements used according to their properties for element attribute including section types. Additional loading on slabs are also added in add mass section of the First Floor and Roof Slabs. Wall loads are also added in the masses of beams.

4.1. ELEMENT ATTRIBUTES

TABLE 3. Section properties of the structural elements

Section Properties	Area (m^2)	$I_{yy}(m^4)$	$I_{yz}(m^4)$	$I_{zz}(m^4)$
COLUMN	0.0135	0.173×10^{-3}	0.237×10^{-19}	0.59×10^{-4}
BEAM (1ST Floor)	0.005428	0.648×10^{-4}	0.13×10^{-19}	0.68×10^{-5}
BEAM (Top Floor)	0.001982	0.812×10^{-5}	0.163×10^{-20}	0.90×10^{-6}
RAFTER	0.2212×10^{-2}	0.228×10^{-5}	0.265×10^{-22}	0.13×10^{-5}
PURLINS	0.523×10^{-3}	0.23×10^{-5}	0.30×10^{-21}	0.190×10^{-6}
TIMBER	0.60×10^{-2}	0.72×10^{-5}	0.66×10^{-22}	0.13×10^{-5}

TABLE 4. Element attributes in ANSY

Section Properties	Real Constant	Element Type	Material Number	Material Section	Section Number
Column	-	1	1	$254 \times 254 \times 107$	1
Beam (1ST Floor)	-	1	1	$254 \times 146 \times 43$	2
Beam (Top Floor)	-	1	1	$152 \times 89 \times 16$	3
Concrete Slab	1	2	2	-	-
Timber	-	1	1	$0.12m$	3
Rafter	-	1	3	$254 \times 254 \times 107$	4
Plywood	2	2	3	-	-
Purlins	-	1	1	C-Section	6
Roof Profile	3	2	1	-	-

5. MODELLING OF THE STRUCTURE

The modeling was started off by changing the orientation of the axis with the z-axis taking the natural y-axis direction. Beam 188 does not take any real constant therefore all section properties of the elements were inputted via the sections tab. Adequate care was taking when messing the elements as it was noticed that the behavior of the structure was not showing the real practical behavior in practice when so much elements were created along each structural element. It was noticed that the whole structure was not deflecting as a unit. The detailed steps carried out during the modeling of the frame structure are detailed in the figures below.

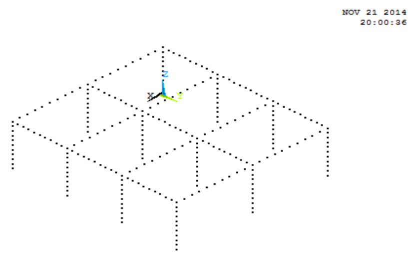


Figure5. Elements generation for the first floor frames

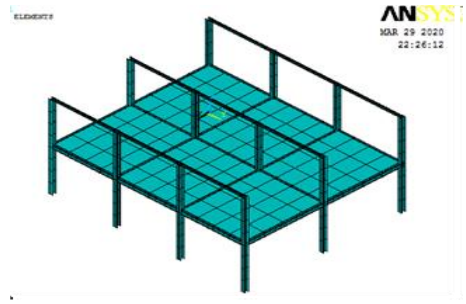
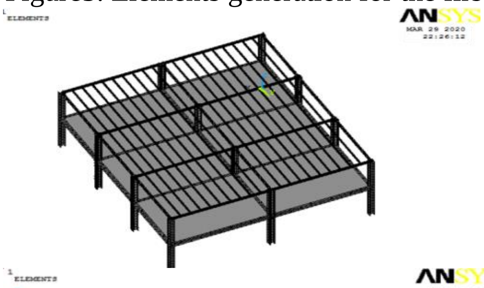


Figure4. Node generation for the first floor



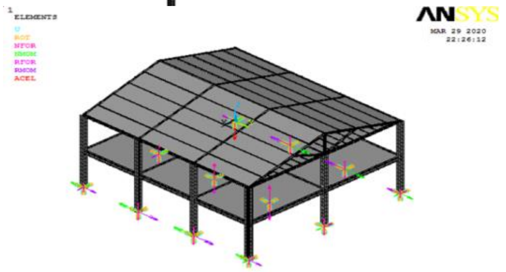


Figure7. Complete Structure with restrain conditions

Figure 6. Roof Elements Generation

5.1. STATIC ANALYSIS

Solution > analysis type > new analysis > static > general postprocessor > list results > reaction solution > select FY

A static structure analysis of the structure is to determine the displacement, stresses and forces in structures or components caused by loads that do not induce significant inertia and damping effects.

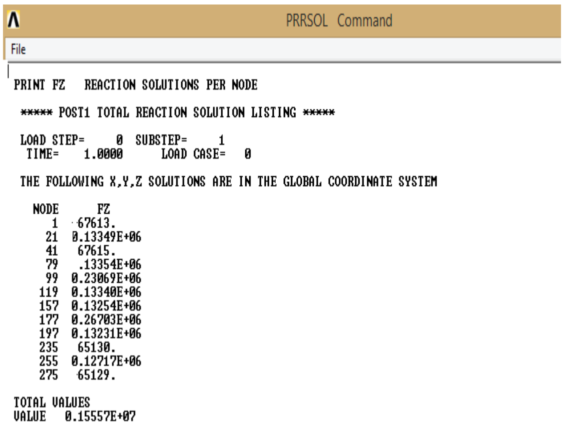
Steady loading and response conditions are assumed; that is the loads and the structure response are assumed to vary slowly with respect to time. Static analysis can however include steady inertial loads (such as gravity and rotational velocity).

To verify the constructed model acceleration due to gravity of 9.81ms (-2) was applied to the structure and the reactions generated from the boundary conditions was checked against the applied load. The results of the static analysis of the reaction forces along the z-axis about the base of the columns are shown below since the orientation of the axis have been changed

Static analysis computes the effects of stable loading situations applied over a certain time. This can happen by avoiding the effect of inertia and damping. It can be used to determine stresses, strain, forces, displacement and mass. For this particular case, it has been used to obtain the total mass of the structure. The results indicate whether the model is constrained adequately and behaving in the expected manner. The results of forces we got from Ansys are converted to kg and then compared to hand calculations as shown below in table.

TABLE 5. Reactions generated from static analysis

NODE NUMBER	F_Z(N)
1	67613
21	133490
41	67615
79	133540
99	230690
119	133400
157	132540
177	267030
197	132310
235	65130
255	127170
275	65129
Total	1555700



PRRSOL Command
File

```
PRINT FZ   REACTION SOLUTIONS PER NODE

 ***** POST1 TOTAL REACTION SOLUTION LISTING *****

 LOAD STEP=     0  SUBSTEP=     1
  TIME=    1.0000      LOAD CASE=   0

 THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEM

   NODE      FZ
      1   67613.
     21  0.13349E+06
     41   67615.
     79   .13354E+06
     99  0.23069E+06
    119  0.13340E+06
    157  0.13254E+06
    177  0.26703E+06
    197  0.13231E+06
    235   65130.
    255  0.12717E+06
    275   65129.

TOTAL VALUES
VALUE   0.15557E+07
```

VALIDATING	TOTAL REACTIONS (KN)
ANSYS	1555.7
HAND CALCULATION	1555.8

The total reaction force in y direction is 1555700 N.

Mass of structure $= \frac{1555700}{9.81} = 158583.08$kg

5.2 VALIDATING THE STRUCTURAL ANALYSIS MODEL

Client: Nuclear Power Station	
BS EN 1998: Design of Structures for Earthquake Resistance BS EN 1991-1-1:2002: Actions on Structures BS EN 1993: Design of Steel Structures	Relevant Building Regulations and Design Codes
Industry (Nuclear) Building to house generators	Intended use of structure
1 hour for all elements	Fire Resistance requirements
Roof: Additional Dead Load=1 kN/m^2 Imposed = 0.75 kN/m^2 Floor: Imposed =3.0 kN/m^2 Internal Partitions & Finishes =1.0 kN/m^2 External Partition (Cladding) = 0.5 kN/m^2	General loading conditions
Density of Steel = 7800 kg/m^3	Material Data

Density of Concrete = 2400 kg/m^2 **Density of Timber Joist (Hardwood) = 570 kg/m^3** **Density of Plywood (Softwood) = 570 kg/m^3**	
All dimension are in millimetres (mm)	**Other Relevant Information**

5.3. GENERAL ARRANGEMENT DRAWINGS

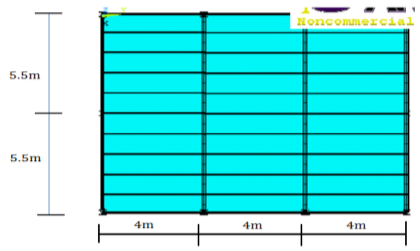


Figure 8. PLAN VIEW OF THE STRUCTURE

Figure9. Elevation View

Estimation of Loads

Top Floor Loads

Dead Loads

Additional Dead Loads	=0.75kN/m^2
No, of Roof Panels	=6
Area of Panel	= 22m^2
Total Dead Load	=99.0kN
Total Dead Load (in kg)	=10091.1kg

Roofing Sheet

Panel Area	= 22 m^2
Number of Panels	=6
Adjusted Thickness	= 0.004 m
Density of Steel	=7800 kg/m^3
Mass of Roofing Sheet	=7800×23.41×0.004×6 =4118.4kg

Purlins
Number of Purlins =11
Cross-section area =0.000523m^2
Length per purlin =12m
Density of Steel =7800 kg/m^3
Mass of Purlins =7800×12×11×0.000523 =538.5kg
Truss
Number of Trusses = 4
Cross-section area = 0.002212m^2
Total of Length per purlin members (Length _TRUSS)
Top Chord =5.85m
Vertical Bracing =2.00m
Diagonal Bracing =2.87m
[Length of TRUSS] =(2×5.85)+2.00+(2×2.87) = 19.44m
Density of Steel =7800 kg/m^3
Mass of Truss Members =7800×19.45×4×0.002212 =1342.1kg
Plywood Floor
Panel Area =22 m^2
Number of Panels =6
Adjusted Thickness = 0.012 m Density of Plywood =720 kg/m^3
Mass of Plywood Floor =720×22×0.012×6 =1140.5kg

Timber Joist
Length = 12m
Number of Joists =18
Cross Sectional Area = 0.0075 m^2
Density of Joist =720 kg/m^3

Mass of Roofing Sheet =720×12×18×0.0075 =1166.4kg

Top Floor Beams
Length of Beam (x-dir) =11m
No, of Beams (x-dir) =4
Length of Beam (y-dir) =12m
No, of Beams (y-dir) =3
Total length of beams =80m
Density of steel =7800kg/m^3
Area of steel beam =0.001982m^2
Mass of Roofing Sheet =80×7800×0.001982 =1236.8kg

Total Dead Load (Mass) for top Floor:

Roofing Sheet = 4118.4kg
Purlins = 538.5kg

Truss Members	= 1342.1kg
Plywood Floor	= 1140.5kg
Timber Joist	= 1166.4kg
Top Floor Beams	= 1236.8kg
Total Floor Dead Load	= 9542.7kg

Imposed Loads

Top Floor

Imposed Loads	$=1kN/m^2$
No, of Panels	=6
Area of Panel	$= 22m^2$
Total Imposed Load	= 132kN
Total Imposed Load (in kg)	= 13455.7kg

Total Top Floor Load

Total Floor Load = D. L+ L . L = 9542.7+13455.7 =22998.4kg

Columns-1st Floor to Roof

Length	= 3m
Number of Columns	=12
Density of steel	$=7800\ kg/m^3$
Area of steel column	$=0.0135\ m^2$

Mass of Columns $=3\times12\times0.0135\times7800=3791$kg

First Floor Loads:

Dead Loads

Partition Dead Loads

Additional Dead Loads	$=1kN/m^2$
No, of Panels	=6
Area of Panel	$= 22m^2$
Total Dead Load	= 132kN
Total Dead Load (in kg)	= 13455.7kg

Cladding

Unit weight of Cladding	$=0.5kN/m^2$
Height of Wall	=3m
Total Length external wall	= 46m
Total Cladding Load	= 69kN
Total Cladding Load (in kg)	=7033.64kg

Concrete Floor

Panel Area	$=22m^2$
Number of Panels	=6
Adjusted Thickness	= 0.2 m

Density of Plywood
Mass of Plywood Floor =2400×22×0.2×6 = 63360kg

Second Floor Beams
Length of Beam (x-dir) =11m
No, of Beams (x-dir) =4
Length of Beam (y-dir) =12m
No, of Beams (y-dir) =3
Total length of beams =80m

Density of steel $=7800kg/m^3$
Area of steel Beam $=0.005428m^2$
Mass of Roofing Sheet =80×0.005428×7800=3387.1kg

Total Dead Load (Mass) for top Floor:
Partition Dead Load = 13455.7kg
Cladding = 7033.64kg
Concrete Floor = 63360kg
First Floor Beams = 3387.1k
Total Floor Dead Load =87236.44kg

Imposed Loads
Top Floor
Imposed Loads $=3kN/m^2$
No, of Panels =6
Area of Panel $= 22m^2$
Total Imposed Load = 396kN
Total Imposed Load (in kg) = 40366.97kg

Total Floor Load
Total Floor Load = D. L+ L. L =87236.44+ 40366.97=127603.41kg

Columns-Ground Floor to 1st Floor
Length = 3m
Number of Columns =12
Density of steel $=7800kg/m^3$
Area of steel column $=0.0135m^2$
Mass of Columns =3×12×0.0135×7800=3791kg

Total Building Load:
Top Floor Load = 22998.4kg
Column (FF to TF) = 3791kg
First Floor Load = 127603.41kg

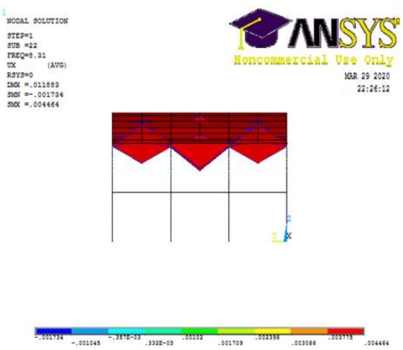
Column (GF to FF) = 3791kg
Total Building Mass =158183.8kg

6. MODAL ANALYSIS

The concept of modal analysis in structural analysis is to determine the natural mode shapes and frequencies of an object or structure during free vibration. It is a common practise to use finite analysis packages to carry out this analysis because the structure being analyse can have arbitrary shape and the results of the analysis will me much more accurate. The equations generated from modal analysis are those seen in Eigen systems. The physical interpretation of the eigenvalues and the eigenvectors which arises from the analysis generate the frequencies and the corresponding mode shapes. The tabulated results below shows the result generated from the ANSYS model and the frequencies with the most participating mass selected across all the Cartesians directions.
A fixed base modal analysis of the model was carried out and the frequency of interest was extracted, in selecting the frequency of interest along 3-Directional axis the mode shapes with the highest participating masses were considered. The no of modes extracted was 50 in numbers and the range of frequency used varies from 0-33Hz, and this is because most excitation takes place between this frequency and a discretize time interval of 0.01s was used and this was sufficient as we captured data at a rate of 100seconds per seconds.

Orientation	Mode	Frequency (HZ)	Effective Mass (KG)	Total Mass (KG)	Cumulative Percentage (%)
X	1	3.59	136771	157374	87
	2	8.31	16326	157374	99.8
Y	1	2.80	147840	157550	93.84
	2	5.95	9698.4	157550	99.99
Z	1	12.68	97009	121286	99.6
	2	12.62	11565.6	121286	70

The figures below shows the deflected shape for the mode shapes obtained from the analysis in ANSYS.



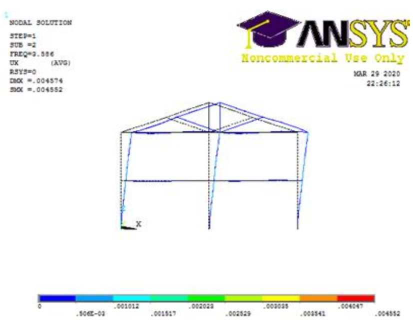
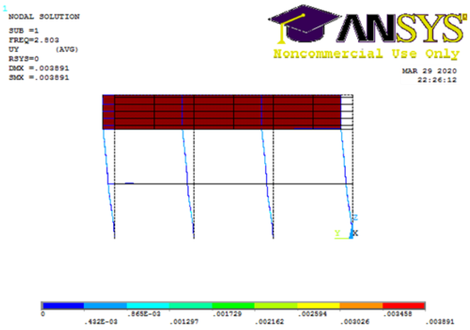


Figure9.Diagram showing Mode 1 X- Direction

Figure10. Diagram showing Mode 2 X-Direction



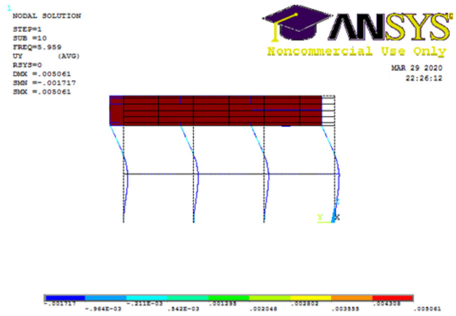
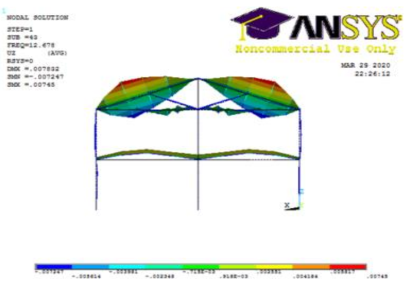


Figure11. Diagram showing Mode 1 Y-direction.

Figure12. Diagram showing Mode 2Y-direction



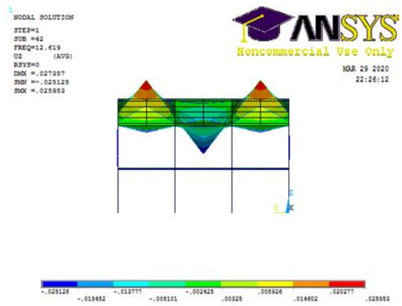


Figure13.Diagram showing Mod 1 Z-direction

Figure14.Diagram showing Mode 2 Z-direction

Modal Analysis

$M_1 = 127849.81$ kg $\qquad M_2 = 22998.4$kg

$E_s = 210000\ \mathrm{N/mm^2} \qquad I_{yy} = 1.751 \times 10^{-4}\ \mathrm{m^4}$

$I_{zz} = 5.928 \times 10^{-5}\ \mathrm{m^4}$ From Blevins Frame 23,

Stiffness in The y-direction.

$$k_1 = \frac{12 \cdot E_s \cdot I_{zz}}{L^3} \cdot N_c$$

$$k_1 = \frac{12 \times 2.1 \times 10^{11} \times 5.928 \times 10^{-5}}{3^3} \times 12 = 66393.6\ \mathrm{N/mm}$$

$$k_1 = k_2 = 66393.6\ \mathrm{N/mm}$$

$$f_1 = \frac{1}{2^{\frac{3}{2}} \cdot \pi} \cdot \left[\frac{k_1}{m_1} + \frac{k_2}{m_1} + \frac{k_2}{m_2} - \left(\left(\frac{k_1}{m_1} + \frac{k_2}{m_1} + \frac{k_2}{m_2}\right)^2 - \frac{4 \cdot k_1 \cdot k_2}{m_1 . m_2}\right)^{0.5}\right]^{0.5}$$

$$f_1 = \frac{1}{2^{\frac{3}{2}} \cdot \pi} \cdot \left[\frac{66393.6}{127849.81} + \frac{66393.6}{127849.81} + \frac{66393.6}{23260.32} - \left(\left(\frac{66393.6}{127849.81} + \frac{66393.6}{127849.81} + \frac{66393.6}{23260.32}\right)^2 - \frac{4 \cdot\ 66393.6 \cdot\ 66393.6}{127849.81 \times 23260.32}\right)^{0.5}\right] = 9.37\ \mathrm{Hz}^{0.5}$$

$$T_1 = \frac{1}{f} = \frac{1}{9.37} \qquad T_1 = 0.11s$$

$$f_2 = \frac{1}{2^{\frac{3}{2}} \cdot \pi} \cdot \left[\frac{k_1}{m_1} + \frac{k_2}{m_1} + \frac{k_2}{m_2} + \left(\left(\frac{k_1}{m_1} + \frac{k_2}{m_1} + \frac{k_2}{m_2}\right)^2 - \frac{4 \cdot k_1 \cdot k_2}{m_1 . m_2}\right)^{0.5}\right]^{0.5}$$

$$f_2 = \frac{1}{2^{\frac{3}{2}} \cdot \pi} \cdot \left[\frac{66393.6}{127849.81} + \frac{66393.6}{127849.81} + \frac{66393.6}{23260.32} - \left(\left(\frac{66393.6}{127849.81} + \frac{66393.6}{127849.81} + \frac{66393.6}{23260.32}\right)^2 - \frac{4 \cdot\ 66393.6 \cdot\ 66393.6}{127849.81 \times 23260.32}\right)^{0.5}\right]^{0.5} = 3.29\ \mathrm{Hz}$$

Mode Shapes and Participating Mass

Mode Shape 1

$$\chi_1 = 1$$

$$\chi_2 = 1 + \frac{k_1}{k_2} - \frac{M_1}{k_2} \cdot (2 \cdot \pi \cdot f_1)^2$$

$$\chi_2 = 1 + \frac{66393.6}{66393.6} - \frac{127849.81}{66393.6} \cdot (2 \times \pi \times 9.37)^2$$

$$\chi_2 = -4.67$$

$$\phi_1 = \frac{\chi_1}{\chi_2} = \frac{1}{-4.67} = -0.21$$

$$f_2 = \frac{1}{2^{\frac{3}{2}} \cdot \pi} \cdot \left[\frac{k_1}{m_1} + \frac{k_2}{m_1} + \frac{k_2}{m_2} + \left(\left(\frac{k_1}{m_1} + \frac{k_2}{m_1} + \frac{k_2}{m_2} \right)^2 - \frac{4 \cdot k_1 \cdot k_2}{m_1 . m_2} \right)^{0.5} \right]^{0.5}$$

$$f_2 = \frac{1}{2^{\frac{3}{2}} \cdot \pi} \cdot \left[\frac{66393.6}{127849.81} + \frac{66393.6}{127849.81} + \frac{66393.6}{23260.32} - \left(\left(\frac{66393.6}{127849.81} + \frac{66393.6}{127849.81} + \frac{66393.6}{23260.32} \right)^2 - \frac{4 \cdot 66393.6 \cdot 66393.6}{127849.81 \times 23260.32} \right)^{0.5} \right]^{0.5}$$

$$f_2 = 3.29 \text{ Hz}$$

$$T_2 = 0.30 s$$

Mode Shapes and Participating Mass

Mode Shape 1

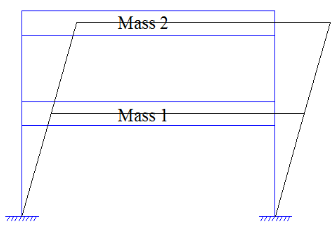
$$\chi_1 = 1$$

$$\chi_2 = 1 + \frac{k_1}{k_2} - \frac{M_1}{k_2} \cdot (2 \cdot \pi \cdot f_1)^2$$

$$\chi_2 = 1 + \frac{66393.6}{66393.6} - \frac{127849.81}{66393.6} \cdot (2 \times \pi \times 9.37)^2 = -4.67$$

$$\phi_1 = \frac{\chi_1}{\chi_2} = \frac{1}{-4.67} = -0.21$$

$$\phi_2 = \frac{\chi_1}{\chi_1} = \frac{1}{1} =$$



Participating Mass for Mode Shape 1 (PM1)
Earthquake Excitation Factor

$$L_1 = M_1 \cdot \phi_1 + M_2 \cdot \phi_2$$

$$L_1 = (127849.81 \times -0.21) + (23260.32 \times 1) = -4100.36 \text{kg}$$

$L_1 = -4100.36\text{kg}$

Modal Mass

$MM_1 = M_1 \cdot \phi_1{}^2 + M_2 \cdot \phi_2{}^2$

$MM_1 = 127849.81 - 0.21^2 + 23260.32 \times \quad 1^2$

$MM_1 = 29115.68\text{kg}$

$$PM_1 = \frac{L_1{}^2}{M_1} = \frac{-4100.36^2}{29115.68} = 577.45\text{kg}$$

$$\%\ \text{of participating mass} = \frac{PM_1}{M_1 + M_2} \times 100$$

$$\%\ \text{of participating mass} = \frac{577.45}{127849.81 + 23260.32} \times 100$$

$\%$ of participating mass $= 0.40\%$

Mode Shape 2

$$\chi_1 = 1$$

$$\chi_2 = 1 + \frac{k_1}{k_2} - \frac{M_1}{k_2} \cdot (2 \cdot \pi \cdot f_2)^2$$

$$\chi_2 = 1 + \frac{66393.6}{66393.6} - \frac{127849.81}{66393.6} \cdot (2 \times \pi \times 3.29)^2$$

$$\chi_2 = 1.18$$

$$\phi_1 = \frac{\chi_1}{\chi_2} = \frac{1}{1.18} = 0.85$$

$$\phi_2 = \frac{\chi_1}{\chi_1} = \frac{1}{1} = 1$$

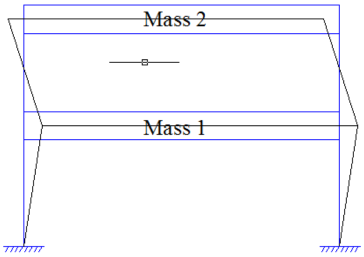
Participating Mass for Mode Shape 2 (PM2)

Earthquake Excitation Factor

$L_2 = M_1 \cdot \phi_1 + M_2 \cdot \phi_2$

$L_2 = (127849.81 \times 0.85) + (23260.32 \times 1)$

$$L_2 = 131950.2\text{kg}$$



Modal Mass

$$MM_1 = M_1 \cdot \phi_1{}^2 + M_2 \cdot \phi_2{}^2$$
$$MM_1 = 127849.81 \times 0.85^2 + 23260.32 \times 1^2$$
$$MM_1 = 115661.6\text{kg}$$
$$PM_2 = \frac{L_2{}^2}{MM_1} \qquad PM_2 = \frac{131950.2^2}{115661.6} = 150532.7\text{kg}$$
$$\%\text{ of participating mass } = \frac{PM_2}{M_1 + M_2}.100$$

$$\%\text{ of participating mass } = \frac{150532.7}{127849.81 + 23260.32}.100$$

$\%$ of participating mass $= 99.62\%$
Dominant Mode Shape = Mode Shape 2

$$f_2 = 3.29\text{ Hz}$$
$$S_e(f_2) = 2.7\text{m/s}^2$$

$$\text{Base Shear}, F_B = 150532.7\text{kg} \times 2.7\text{m/s}^2 = 406.4\text{kN}$$

$$\text{Lateral Force at Floor}, F_i = F_B \cdot \frac{z_k m_k}{\sum_j z_j m_j}$$
$$F1 = 406.4 \cdot \frac{3 \times 127849.81}{(3 \times 127849.81) + (6 \times 23260.32)} = 298\text{Kn}$$

$$F_2 = 698.84 \cdot \frac{6 \times 23260.32}{(3 \times 127849.81) + (6 \times 23260.32)} = 108.43\text{kN}$$

Overturning Moment, Mo

$$M_o = F_1 \times z_1 + F_2 \times z_2$$
$$M_o = 298 \times 3 + 108.43 \times 6 = 1544.62\text{kNm}$$

Restoring Moment, MR
$M_R = (M_1 + M_2) \times A \times (9.81 - 3.90)$

$$M_R = (127849.81 + 23260.32) \times 5.5 \times (9.81 - 3.90)$$
$$M_R = 8153.2\text{kNm}$$

The structure is stable since Restoring Moment is greater The Overturning Moment

Mode shapes	ANSYS (Hz)	Hand Calculation (Hz)
X-Direction	3.59	3.29
Y-Direction	2.80	-
Z-Direction	12.68	9.37

7. Discussion

The main objective of this assignment was to understand the structural behavior of the building under seismic loading. The building was modeled through finite elements to achieve better and more accurate results. Both the floors and roof were divided into finite elements, as subdividing of panels into finer elements will result in more precise values.

There are slight variations in the results obtained in ANSYS analysis package when compared to the hand calculations obtained using Blevins simplified analysis approach. The differences in the values obtained could be as results of the various assumptions made by Blevins. Before considering the various assumptions made by Blevins it is worth mentioning that since Blevins is a simplified analysis approach its analysis is 2-Dimensional therefore it doesn't considers the Z-directional axis in analysis. It should be noted as mentioned earlier that in modeling this building the Z-axis was replaced by the conventional Y-Directional axis. One of the assumptions made by Blevins is that the columns are considered to be weightless that is while computing the frequencies in Blevins the self-weight of the columns in the structures was not considered while ANSYS considered this while analysing the structure.

Blevins assumed the deformation of the structure in both directions as equal therefore analysing the deformation to be linear along the path of bending while in ANSYS it is analyses as a curved deformed shape thereby causing differences in the values of the deformed shapes computed along both directions considered.

8. Conclusions

1. All the analysis were carried out appropriately and desired results were obtained. The ANSYS results matched the Hand Calculations, but still improvements can be done.
2. As mentioned in discussion, that changes to meshing and boundary conditions could results in much results.
3. ANSYS is fast way to analyses the most complicated structures.
4. If the modeling is done well, ANSYS will give more accurate results.
5. The graphs and data produced by this software can help out the design engineers

As from ANSYS we can predict the future behavior of structure under different seismic loading so it can be very good helping tool for the design engineers for seismic design.

References

[1]. Steel Designer's manual, 6th Edition, Blackwell publishing Ltd, London2007B. Davidson& W. Graham

[2]. Taucer, F.F., Spacone, E, and Filippou, F.C., 1991. "A fiber beam-column element for seismic response analysis of reinforced concrete structures", Report No.UCB/EERC-91/17, Earthquake Engineering Research Center College of engineering, University of California, Berkeley

[3]. M.R. Shehata, G.A. Al-Saadi, H. Abou-Elfath, E.A. El-Hout, Push over static analysis of moment resisting steel frames, in: Sixth Alexandria International Conference on Structural and Geotechnical Engineering, AICSGE 6, Alexandria, Egypt, 2007, pp. ST133–ST149.

[4]. Seismic Design of Buildings to Euro codes, 1st Edition, Spon Press Ltd 2009 Ahmed Y. Eghazouli

[5]. Steel Building Design: Worked Examples – Open Sections. Steel Construction Institute 2009.

[6]. P A Kirby & D A Nethercott, Design for Structural Stability, Constrado Monographs, 1979

[7]. M.A.A. El-Shaer, Effect of earthquake on steel frames with Partial rigid connection, J. Eng. Sci., Assiut Univ. 40 (2) (2012) 343–352.

[8]. Vecchio FJ, Collins MP. The modified compression field theory for reinforced concrete elements subjected to shear. ACI J Proc 1986;83(6):219–31.

[9]. Bentz EC. Explaining the riddle of tension stiffening models for shear panel experiments. J Struct Eng., ASCE 2005;131(9):1422–5.

[10]. Stramandinoli RSB, Rovere HLL. [6] Vecchio FJ, Collins MP. The modified compression field theory for reinforced Eng. Struct 2008;30:2069–80