

Seismic Vulnerability Analysis of Airport Traffic Control Towers

Sassan Eshghi¹ and Hooman Farrokhi²

1. Assistant Professor, International Institute of Earthquake Engineering and Seismology (IIIES), email: s.eshghi@dena.iiies.ac.ir
2. Structural Engineer, Parsab Structural Consulting Co., email: hoomanfarrokhi@yahoo.com

ABSTRACT: *Airport Traffic Control Towers (ATCT's) are among the most critical structures in an airport that are expected to keep their level of serviceability during and after severe disasters like strong earthquakes. Seismic vulnerability of these structures is the matter of great importance during immense ground motions due to their high sensitiveness to structural and non-structural damage. On the other hand, few of these towers are constructed using special structural systems compared to the ordinary residential and industrial structures, because of their uncommon topology and expected function. Therefore analysis and design of these structures cannot be performed using common building codes and methods, but needs a detailed investigation on the seismic behavior of the structure based on the geotechnical characteristics of its site. The control tower under the study is a reinforced concrete structure consisting of four symmetric flexural flanges connected with floor slabs in twelve elevations. The dynamic analysis of the structure is performed by ANSYS finite element program. The lateral forces are estimated performing spectral analysis. The ultimate strength of the structure and the cracking patterns are revealed through a nonlinear static push-over analysis.*

Keywords: Control tower; Tower structure; Reinforced concrete; Material modeling of reinforced concrete; Vulnerability analysis; Nonlinear analysis; Dynamic analysis; Push-over analysis

1. Introduction

Airports are among the most important urban infrastructures that should keep their serviceability during and after severe earthquakes in order to manage their crucial workload and critical role.

One of the most important facilities in an airport with a direct influence on the serviceability is the traffic control tower (ATCT). To organize the air traffic, each airport is usually served by one or more control tower(s). Therefore, no landing or take off may take place if the control tower is shut down and the whole functionality of the airport will be halted temporarily. If the control tower cannot operate properly, due to structural or non-structural damages during earthquakes, the airport

may experience some long-term disabilities. For example during 1964 Alaska earthquake, the control tower of the Anchorage International Airport, experienced severe structural and nonstructural damages, and due to these damages, the airport was closed for more than a month [9]. Also Tacoma-Seattle International Airport was halted for a half-day period due to significant non-structural damage to the control tower, caused during recent earthquake of Seattle, on 28 February 2001. Although the damage to the control tower was minor, the control tower lost its functionality and the airport reopened using a portable controlling facility with a very limited capacity [7].

ATCT(s) usually contain many sensitive electronic equipment that should be treated as secondary structures which are considered to sustain strong motions. Also uncontrolled vibrations of the control room may cause some negative mental effects on the controlling staff in the form of anxiety, fright, or even panic that may temporarily reduce their level of operationally [8].

The control tower shown in Figure (1) is a typical reinforced concrete airport control tower that is regionally designed according to the previous seismic codes. A seismic vulnerability analysis of control towers in highly seismic regions, using modified



Figure 1. A typical control tower.

design philosophies and detailed seismic and geotechnical investigations are urgently needed. Some of the reasons can be mentioned as follows:

1. Due to special structural system of the tower, an appropriate design procedure should be considered for the analysis which can be significantly different from the conventional design methods recommended by seismic design codes.
2. The finite element software used in analysis of the structure should have the capability to simulate possible nonlinear behaviour which may take place at the structure. Most Finite element packages are based on many simplifying assumptions that may cause significant deviations from actual behavior of the structure. The behavior of the tower is rather

complicated compared to ordinary structural systems.

3. Many control towers were analysed using a linear structural analysis software. Slabs are considered to act like rigid diaphragms and structural mass is considered lumped at slab levels. These assumptions have direct influence on the whole process of structural analysis of the tower. Slabs are not stiff enough (compared to wings) and wings are much more massive than the slabs. Therefore lumped mass approximation for this type of structure could be far from the reality.
4. Concrete is highly sensitive to three-dimensional stress states and can behave brittle or significantly ductile in different confined or unconfined states of stress. Therefore, in structures that the stress state can vary significantly (e.g. shell structures), the behavior of concrete can also be considerably variable. Cracking effects are not considered in the analysis of the structure, and because of this assumption the internal forces due to seismic loading are underestimated in the tower. As it will be shown, slabs can be cracked very rapidly and will lose the great part of their stiffness at a low level of lateral stimulation. Therefore, the distribution of moments and shear forces will completely alter. This alteration may be neglected by the designer. It should be noted that, simplified techniques used to simulate cracking effect in ordinary reinforced concrete moment frames, (e.g. stiffness reduction factors, advised in concrete design codes) can not be used in this type of structures. Nonlinear behavior of reinforced concrete and cracking patterns in the tower structures are much more complicated than the ordinary framed reinforced concrete structures.
5. Many towers are originally analyzed using equivalent static seismic load based on national seismic codes. Control towers are not categorized as a common symmetric structural system and due to its special conditions, equivalent static lateral loading may not support the conservative design philosophy.
6. When using equivalent static loading combined with a linear analysis method, a performance factor named "*R*" should be considered for the whole structure to reduce the seismic loads imposed on the structure at a severe earthquake

to include nonlinear deformations. “ R ” factor used to be considered to “4” for designing control towers. Up to this date, no acceptable document recommends such an R factor for this type of structure for including nonlinear deformations. Considering a certain performance factor for this kind of structure may not lead to a conservative result.

Based on the above mentioned reasons, a rigorous nonlinear analysis of control towers are recommended, since common methods may end to a drastic deviation from real behaviour of the structure. The material modeling is one of the most important and sensitive factors in nonlinear analysis of concrete structures and may affect the outcome results significantly.

2. The Control Tower

A reinforced concrete tower is considered for investigation in this study, as shown in Figure (2). This tower is approximately 62 meters high and its approximate horizontal width is about 12.00 meters in two directions. It consists of four symmetric box wings which are connected with floor diaphragms in 12 levels. This section is constant from bottom to top of the tower. A control room is located at the top of the tower as shown in Figure (3), which is a lightweight steel frame in two stories that homes the controlling staff and equipment. The wings are hollow boxes with some openings (doors) having 30cm thickness. The empty spaces inside the wings are used for architectural systems such as elevators, stairways, store-rooms, etc. According to Iranian seismic design code, the soil of the airport site is classified as type *II*. The reinforcement of the wings consists of three sets of reinforcing bars: a longitudinal set (vertical), a hoop set in two layers (horizontal), and ties between the hoop bars. A typical section of the wings reinforcement is shown in Figure (4). The floor diaphragms, which are reinforced concrete slabs, are in the form of octagon that are connected to the wings in four sides. The thickness of the slabs is 25cm at almost all levels with two layers of reinforcement in two perpendicular horizontal directions and is connected together with ties. The only different slab type is the control room floor, which is a 30cm slab with two flexural beams in each direction to produce sufficient stiffness. The foundation of the tower is a mat foundation in the form of an octagon with

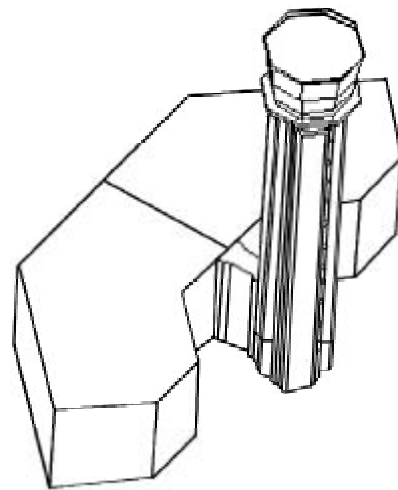


Figure 2. 3D view of the tower.

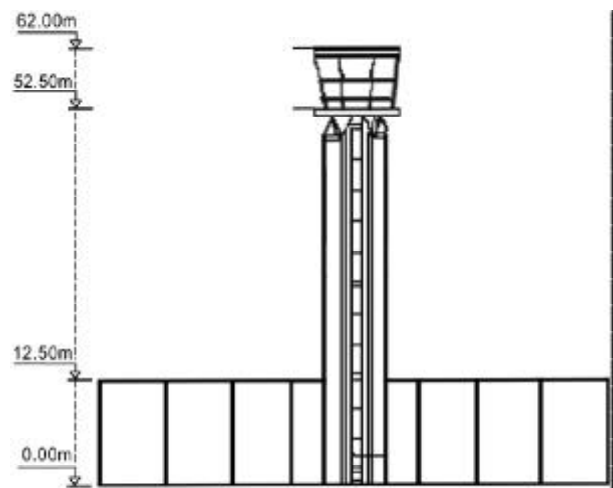


Figure 3. Tower elevation view.

approximately 16 meters width and 1 meter thickness. Considering sufficient stiffness for the mat foundation to act rigid, it is assumed that the tower shaft is fixed at the bottom. The form of the tower and typical reinforcement pattern of the wings are shown in Figure (4).

3. Finite Element Modeling of the Tower

A finite element model of the tower is adopted which is shown in Figure (5). The analysis is performed by the means of the *ANSYS* finite element program, using *SOLID65*, 8-node brick element illustrated in Figure (6) that is consistent with a material model for reinforced concrete in *ANSYS*, see Figure (7). The material model presented for the concrete is a plasticity-based nonlinear material model introduced in the following paragraph.

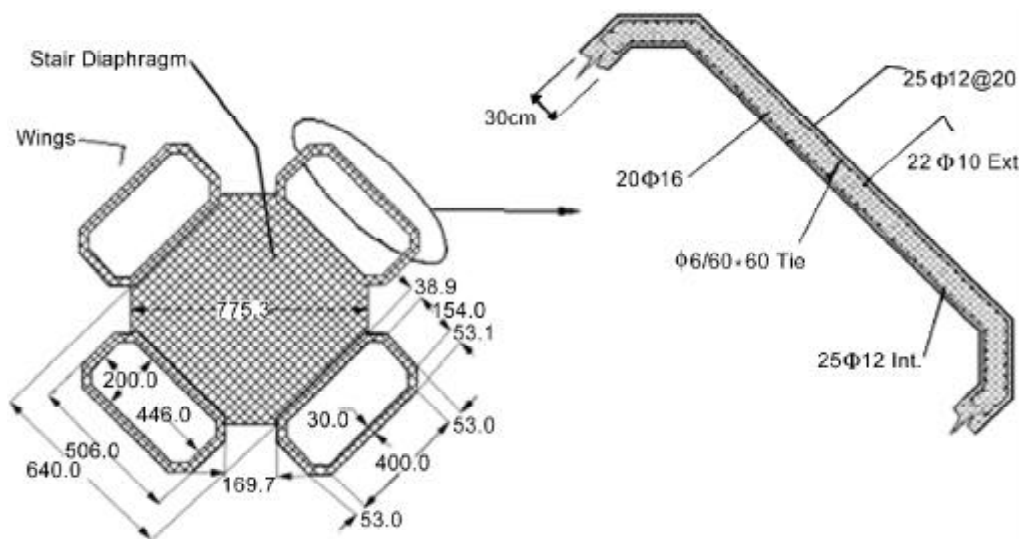


Figure 4. Tower section and typical reinforcement pattern of the wings (elv =+2.00 m).

The wings are modeled by applying the *ANSYS* extrusion technique and are sliced in the roof levels in order to have shared nodes with the slabs. The mass of the control room is simulated with concentrated masses at the top level. The nodes within a defined tolerance are merged to each other to prevent probable discontinuous edges in the model and the foundation nodes are restrained in all degrees of freedom to simulate fixed support for the tower.

4. Concrete Material Modeling

In this study basic concepts of material modeling of reinforced concrete are reviewed. Then, a material model for reinforced concrete presented in *ANSYS* finite element program is used in the analysis.

The main objectives in the material modeling of the reinforced concrete are to:

1. Predict cracking and crushing in the concrete (using a failure criterion) and to impose the necessary modifications in stiffness matrix (using a cracking approach).
2. Relate stresses with strains in a suitable way that can predict pre-yielding and post-yielding behavior of concrete (done by loading function) [4].

As it is shown in Figure (8), the concrete acts as a linear elastic material under low compressive pressure and almost in the tensile region of stress space. Significant nonlinearity is observed when confining pressure becomes rather moderate or high [2]. In this region failure mechanism of concrete is

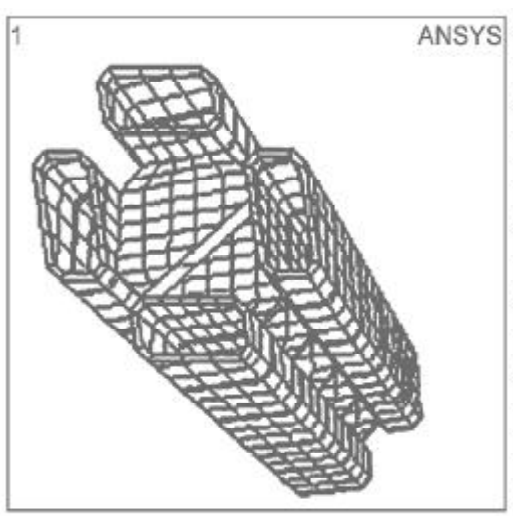


Figure 5. A Finite element model of the tower.

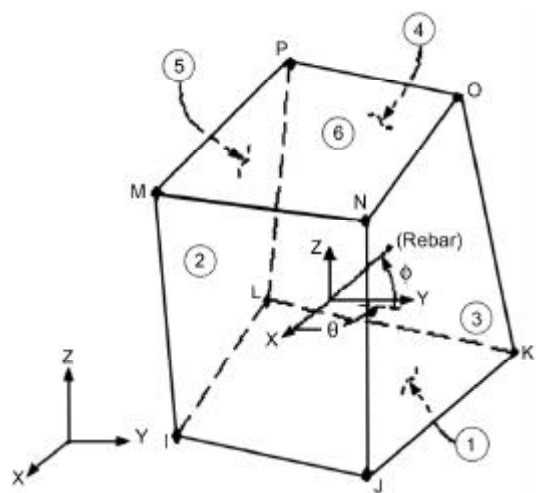


Figure 6. Solid65 element in the ANSYS program for concrete [1].

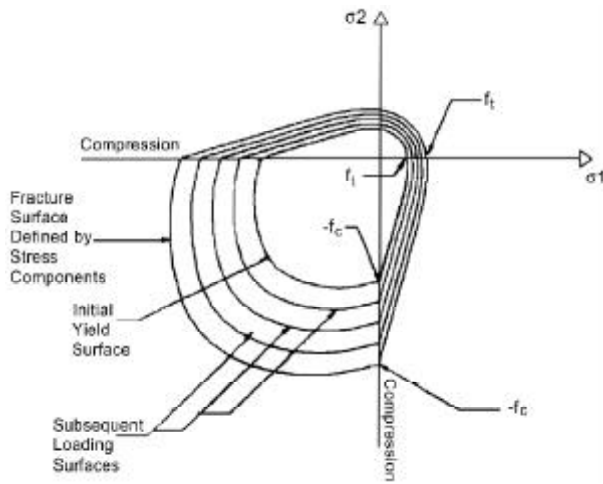


Figure 7. 3D failure surface of concrete.

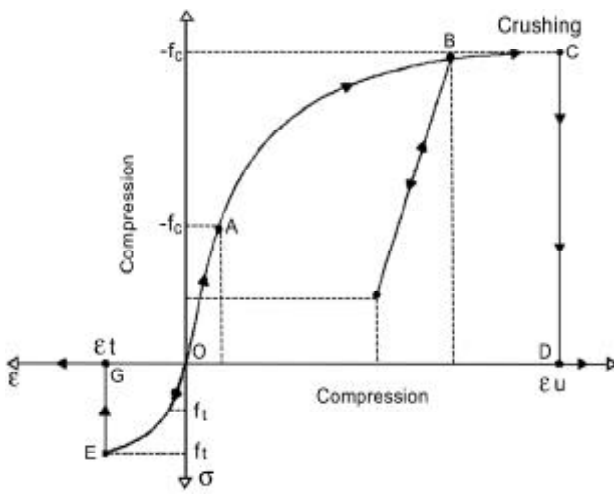


Figure 8. Ideal uni-axial stress strain curve [2].

altered from cleavage to yielding mode, which means that concrete acts like a brittle material in the tensile region and low confining or non-confining pressures and it acts rather ductile in moderate or high confined pressure [4]. Concrete is usually considered linearly elastic material in the tensile region. (Exact observations show that it behaves nonlinearly to some extent) [4]. Among the many time-dependent and independent phenomena that contribute to the nonlinear behavior of the concrete, the cracking and the nonlinear behavior of the reinforcement might be considered as the most important ones. The behavior of the concrete in multi-axial stress greatly depends on the stress states. So, the loading functions are usually defined to predict the triaxial behavior of the concrete in the form of a 3 dimensional surface (hyper-surface if anisotropy is considered) in the 3 dimensional principal stress space [3]. In the

material model for reinforced concrete presented in ANSYS finite element program, a five parameter Willam-Warnke failure criterion is implemented to measure cracking or crushing of the concrete. Also a Drucker-Prager loading function is used to calculate multi axial effects of the stress. A fixed orthogonal smeared crack method is used to impose necessary modifications in the stiffness matrix. Willam-Warnke failure criterion is defined based on five parameters as follows [1]:

1. Maximum Uniaxial tensile stress
2. Maximum Uniaxial compressive stress
3. Maximum biaxial compressive stress
4. Maximum biaxial confined pressure stress
5. Maximum biaxial stress for uniaxial pressure

We can now define functions of failure criterion (S) and a loading function (F) on the basis of the principal stresses and parameters above, but first few more variables should be defined:

$$\cos \zeta = \frac{2\sigma_1 - \sigma_2 - \sigma_3}{\sqrt{2} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]^{1/2}}$$

$$r_1 = a_0 + a_1 \xi + a_2 \xi^2$$

$$r_2 = b_0 + b_1 \xi + b_2 \xi^2$$

$$\xi = \frac{\sigma_h}{f_c}$$

When σ_h is hydrostatic stress

$$\sigma_h = \frac{1}{3}(\sigma_1 + \sigma_2 + \sigma_3)$$

In the equation above $\sigma_1, \sigma_2, \sigma_3$ are the principal stresses and σ_1 is the maximum principal stress and σ_2 is the minimum principal stress (considering positive for tensile stress). Also a_0, a_1, a_2 parameters can be defined with the following equation

$$\left\{ \begin{array}{l} \frac{F_1}{f_c} \left(\begin{array}{l} \sigma_1 = f_t \\ \sigma_2 = \sigma_3 = 0 \end{array} \right) \\ \frac{F_1}{f_c} \left(\begin{array}{l} \sigma_1 = 0 \\ \sigma_2 = \sigma_3 = -f_{cb} \end{array} \right) \\ \frac{F_1}{f_c} \left(\begin{array}{l} \sigma_1 = -\sigma_h^a \\ \sigma_2 = \sigma_h^a - f_1 \\ \sigma_3 = -\sigma_h^a - f_1 \end{array} \right) \end{array} \right\} = \begin{bmatrix} 1 & \xi_t & \xi_t^2 \\ 1 & \xi_{cb} & \xi_{cb}^2 \\ 1 & \xi_1 & \xi_1^2 \end{bmatrix} \begin{bmatrix} a_0 \\ a_1 \\ a_2 \end{bmatrix}$$

Where ξ_1, ξ_{cb}, ξ_t are defined with the following equations

$$\xi_1 = \frac{f_t}{3f_c}; \xi_{cb} = \frac{-2f_{cb}}{3f_c}; \xi_1 = -\frac{\sigma_h^a}{f_c} - \frac{2f_1}{3f_c}$$

And b_0, b_1, b_2 parameters are defined with the following matrix equation

$$\begin{Bmatrix} \frac{F_1}{f_c} \begin{pmatrix} \sigma_1 = \sigma_2 = 0 \\ \sigma_3 = -f \end{pmatrix} \\ \frac{F_1}{f_c} \begin{pmatrix} \sigma_1 = \sigma_2 = -\frac{\sigma_h^2}{f_c} \\ \sigma_3 = -\sigma_h^a - \frac{f_2}{f_c} \end{pmatrix} \\ 0 \end{Bmatrix} = \begin{bmatrix} 1 & -\frac{1}{3} & \frac{1}{q} \\ 1 & \xi_2 & \xi_2^2 \\ 1 & \xi_0 & \xi_0^2 \end{bmatrix} \begin{bmatrix} b_0 \\ b_1 \\ b_2 \end{bmatrix}$$

Where ξ_2 is

$$\xi_2 = \frac{-\sigma_h^a}{f_c} - \frac{f_2}{3f_c}$$

And ξ_0 is the positive root of the following equation

$$r_2(\xi_0) = a_0 + a_1\xi_0 + a_2\xi_0^2 = 0$$

Using this approach, the function F and S are simply defined in four different stress regions and the mathematical form of these functions can be seen in Table (1).

Uncracked concrete stiffness matrix is similar to that of any other elastic material. When the maximum principal stress (σ_1) exceeds maximum tensile stress of the concrete, a crack plane will form, the normal stiffness component to the crack plane is released, and the shear components of the crack plane are reduced. To some extents when the crack closes, the normal component is completely resumed (concrete is considered to be completely healed) and shear components will grow to a significant value [2, 3, 4].

5. Seismic Analysis of the Control Tower

Firstly, the finite element model of the tower is generated using the *SOLID65* brick element available in *ANSYS* program. The nodes connecting the tower shaft with its foundation are restrained in all degrees of freedom (fixed support) and the control room is assumed to have no significant interaction with the structure and considered as a lumped mass at the top level of the shaft. The reinforcement is modeled in three orthogonal axes with volume ratios.

The modulus of elasticity, yield stress and some other mechanical parameters for structural concrete and reinforcement are shown in Table (2).

The outline of the seismic analyses on the

Table 1. Loading functions and failure criterion.

Stress Region	Loading Function (Drucker-Prager)	Failure Criterion (Willam-Warneke)
$0 > \sigma_1 > \sigma_2 > \sigma_3$	$F_1 = \frac{1}{\sqrt{15}} [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]^{\frac{1}{2}}$	$S_1 = \frac{2r_2(r_2^2 - r_1^2) \cos \zeta + r_2(2r_1 - r_2)}{4(r_2^2 - r_1^2) \cos^2 \zeta + (r_2 - 2r_1)^2} \times \frac{[4(r_2^2 - r_1^2) \cos^2 \zeta + 5r_1^2 - 4r_1r_2]^{\frac{1}{2}}}{4(r_2^2 - r_1^2) \cos^2 \zeta + (r_2 - 2r_1)^2}$
$\sigma_1 > 0 > \sigma_2 > \sigma_3$	$F_2 = \frac{1}{\sqrt{15}} [(\sigma_2 - \sigma_3)^2 + \sigma_2^2 - \sigma_3^2]^{\frac{1}{2}}$	$S_2 = \frac{\left(1 - \frac{\sigma_t}{f_t}\right) 2P_2(P_2^2 - P_1^2) \cos \zeta + P_2(2P_1 - P_2)}{4(P_2^2 - P_1^2) \cos^2 \zeta + (P_2 - 2P_1)^2} \times \frac{[4(P_2^2 - P_1^2) \cos^2 \zeta + 5P_1^2 - 4P_1P_2]^{\frac{1}{2}}}{4(P_2^2 - P_1^2) \cos^2 \zeta + (P_2 - 2P_1)^2}$
$\sigma_1 > \sigma_2 > 0 > \sigma_3$	$F_3 = \sigma_i \quad i = 1, 2$	$S'_3 = \frac{f_t}{f_c} \left(1 + \frac{\sigma_3}{S_2(\sigma_i, 0, \sigma_3)}\right) ; \quad i = 1, 2$
$\sigma_1 > \sigma_2 > \sigma_3 > 0$	$F_4 = \sigma_i \quad ; \quad i = 1, 2, 3$	$S_4 = \frac{f_t}{f_c}$

control tower is explained below:

1. A modal analysis is performed, to reveal the vibration mode shapes and natural frequencies. These results will be used later by the a mode superposition analysis.
2. A spectral analysis is performed using the modal analysis output results. This analysis will combine the responses of the structure in different vibration modes using *SRSS* combination method using the standard acceleration response spectra recommended by the Iranian code for seismic design, see Figure (9).
3. A push-over analysis is performed, which is a nonlinear static analysis with the main objective of tracing the failure mechanisms in the structure and finding the ultimate strength of the structure. In this analysis, a concentrated

load is located at the top of the tower and the response of the structure is then followed. Finally the relation between the load and deflection at the top of the structure is carried out.

5.1. Modal Analysis

A modal analysis was performed using Block-Zancos mode extraction method which uses less computer resources with an acceptable level of accuracy. Typically most of the effective mass of the structure is covered in the first translation mode (75%). Some basic facts about the first 10 vibration modes and their natural frequencies of the tower structure are shown in Table (3) and Figure (10).

Table 2. Mechanical parameters of the materials used in the analyses of the tower.

Parameter	Structural Concrete	Structural Steel
Material Type	B-300	A III
Modulus of Elasticity	2.66E+10	2.06E+11
Mean Poison's Ratio	0.20	0.30
Maximum Compressive Strength	2.40E+07	N.A (*)
Maximum Tensile Strength	2.75E+06	3.92E+08
Density	2.40E+07	7.70E+04

Table 3. Vibration modes of the tower.

Mode #	Natural Frequency	Form of vibration
1	1.2095	Simple Translation(Y)
2	1.2095	Simple Translation(Y)
3	3.6266	1 st Mode Torsion
4	5.1583	2 nd Mode Translation (Y)
5	5.1584	2 nd Mode Translation (X)
6	10.572	2 nd Mode Torsion
7	12.2	3 rd Mode Translation (Y)
8	12.20	3 rd Mode Translation (X)
9	13.739	1 st Vertical Mode
10	17.790	2 nd Vertical Mode

(*)- -Not Applicable
 "Parameters in Table (2) are presented in SI units."

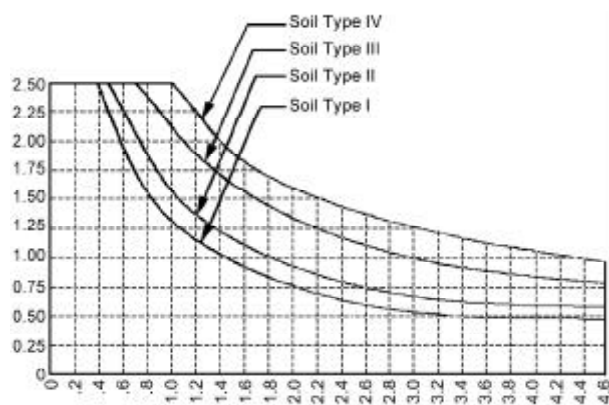


Figure 9. Response spectra used in the spectrum analysis [10].

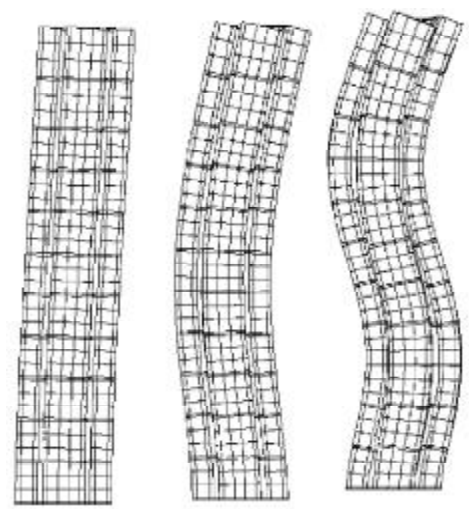


Figure 10. 1st, 2nd, and 3rd translation horizontal vibration modes.

5.2. Spectral Analysis

The seismic vulnerability analysis of typical control tower, based on a spectral analysis, is performed according to the existing Iranian seismic code.

After performing the modal analysis, the output results were used to find the dynamic mass participation factors.

Considering the response spectra advised by the Iranian seismic code of practice and imposing different magnification factors, a mode superposition analysis has been performed and stresses was checked at critical regions of the structure. In addition, shear distribution vertical can be graphed as shown in Figure (11) and accessed for further analyses on the tower. The soil of the region was considered type *II* according to the Iranian code of seismic design. Some other assumptions in the spectrum analysis are [10]:

$$V = C.W$$

$$C = A.B.I/R$$

$$A = 0.35 \quad \text{Normalized acceleration}$$

$$I = 1.20$$

Importance factor

$$R = 1$$

Performance factor

$$T_o = 0.50$$

Assumed natural frequency of soil

(W) is the total seismic weight of the structure and (V) is the base shear force.

Taking the first 3 horizontal vibration modes for each direction, approximately 96 percent of the effective seismic translational mass in that direction is covered. Due to the symmetric form of the structure in plan and elevation, the vibration modes are simply distinguished and are independent in two the horizontal perpendicular directions. It then seems moral to analyze the structure in one horizontal direction independently. The response of the structure was combined in the first 3 horizontal modes using *SRSS* mode combination method and the final results was combined with the gravity dead and live loads using following load combinations:

$$LC1: 1.40D + 1.70L$$

$$LC2: 1.20D + 1.2L + 1.2EQ$$

$$LC3: 1.0D + 1.4EQ$$

Where D stands for Dead load; L stands for Live load; and EQ stands for Earthquake load. The second load combination was almost the critical one and the output results of this combination are plotted and discussed in the next section.

5.3. Push-Over Analysis

Concentrated loads are located at the top of the structure and with increasing these loads, the cracking in the tower is traced. The first cracks initiate in the floor slabs at a very small proportion of the ultimate lateral displacement capacity of the tower and broaden quickly with the increase of the lateral load. Then the first tensile cracks appear in the outermost edge of the tensile wing and propagate through the compressive region, see Figure (12). Finally, the failure of the tower will take place because of the structural failure due to propagation of tensile cracks and formation of a broad plastic region, see Figure (13). The load deflection curve for the top level of the tower is plotted in Figure (14), which is generally consisted of three different regions as:

- i) The first region illustrates an elastic region for the structure. The diagram has a linear form with a constant slope in this region and no significant cracking has been taken place in the structure.
- ii) In the second part of the diagram, some nonlinearities are observed and the stiffness is reduced (reduction in the slope of the curve),

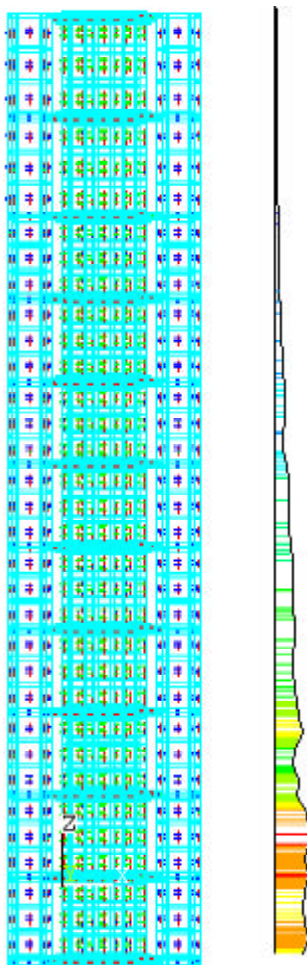


Figure 11. Distribution of shear force along the height of the control tower based on spectral analysis.

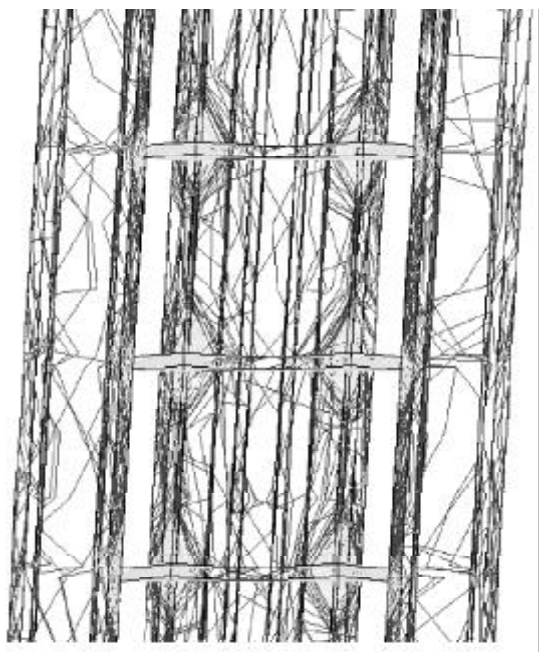


Figure 12. Local stress distribution near the edges of slabs in spectrum analysis.

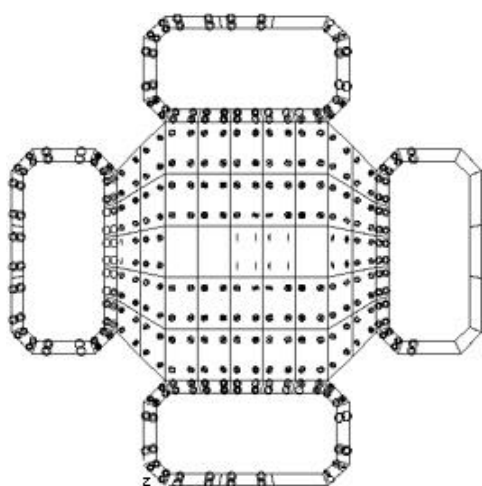


Figure 13. Crack patterns in ultimate state in plan.

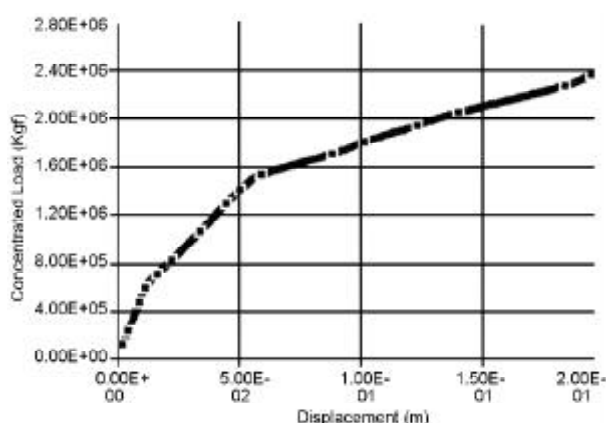


Figure 14. Load displacement curve of the tower.

which is due to cracks formed in the slabs.

- iii) In the third part of the curve, cracks will initiate in the wings, therefore the slope of the curve is greatly decreased and finally the ultimate strength of the structure is reached.

6. Conclusions

A seismic vulnerability analysis of a typical airport control tower has been performed based on the current Iranian code for seismic design. According to the performed analyses and the existing outcome results, the following conclusions are pointed out:

- ❖ ANSYS material model for reinforced concrete uses Drucker-Prager loading function, which is not suitable for the modeling of the shear hysteresis behavior of the concrete [2]. It also uses Willam-Warnke failure criterion and because of the different functions for failure criterion and loading function, some instability in the material response is predictable.
- ❖ The stress contours of the spectrum analysis reveals that the slabs are the most critical regions of the structure in comparison to the wings. While the stress in box wings does not go further from a tiny proportion of the ultimate stress, the slabs suffer from broad cracking patterns and critical stress states.
- ❖ The cracking pattern in the push over analysis shows that the floor diaphragms do not have sufficient stiffness to connect the wings together properly, therefore wing responds to lateral loads individually. This can be estimated due to nondependent cracking pattern in each wing of the tower. By increasing the stiffness of slabs (increasing thickness) the maximum deformation capacity of the structure can be greatly modified.
- ❖ It is clarified that slabs are so much flexible, compared to box wings of the tower, and they lose their integrity due to the large rotations at the corners. Therefore, it seems unwise to consider them as rigid diaphragms which constraint side wings together at specific elevations. In addition, with respect to the geometrical dimensions of the structure, it is revealed that slabs are not so massive in comparison with side wings of the tower and a large part of the tower mass is spread along wings. Therefore, lumped mass approximation is not a proper assumption for seismic analysis of this kind of structure. According to the

technical documents of the control tower, slabs are considered to be rigid diaphragms and seismic mass of the structure is considered lumped at slab levels. These two poor considerations may develop significant errors in the outcome results.

Acknowledgement

The authors would appreciate the assistance of Ms. Yasamin O. Izadkhah for proof-reading and Mrs. Maryam Khaledi for help in preparing the figures of the manuscript.

References

1. ANSYS. References and Manuals: Basic Analysis Procedures Guide “ANSYS Advanced Analysis Techniques Guide-ANSYS Operation Guide-ANSYS Modeling & Meshing Guide-ANSYS Structural Analysis Guide- ANSYS Commands Reference-ANSYS Elements Reference-ANSYS Theory Reference ANSYS V.5. 40_ ANSYS Co. (1997), 9th Edition SAS IP Inc.©
2. Chen, W.F. (1982). “*Plasticity in Reinforced Concrete*”, McGraw Hill Book Company.
3. Chen, W.F. and Haan, D.J. (1988). “*Plasticity for Structural Engineers*”, Springer-Verlag Book Co.
4. Chen, W.F. (1993). “Concrete Plasticity, Macro & Micro Approaches”, *International Journal of Mechanical Sciences*, **35**(12), 1097-1109.
5. Chopra, A.K. (1995). “*Dynamics of Structures*”, Prentice-Hall.
6. Neville, A.M. (1981). “*Properties of Concrete*”, Pitman Publishing INC.
7. Pierepiekarz, M.R. , Ballantyne, D.B. and Hambutrger, R.O. (2001). “Damage Report from Seattle”, Civil Engineering Magazine.
8. Roark, M., Turner, K.Z., and Gould, P.L. (2000). “Seismic Vulnerability of Airport Facilities”, *Proceedings of 12th World Conf. on Earthquake Engineering*, New Zealand, Oakland, Paper No. 0003.
9. <http://www.nisee.org>
10. “Iranian Code of Practice for Seismic Resistant Design of Buildings”, (1997). Standard No. 2800, Second Edition.