Structural performance of a steel tower model under seismic loading

Muaid Adnan¹ and Saad Al-Wazni²
¹PhD, Lecturer at Dep. of Civil Eng., Faculty of Eng., University of Kufa, Iraq.
²PhD, Assist. Prof. at Dep. of Civil Eng., Faculty of Eng., University of Kufa, Iraq.

Corresponding author: saad.alwazni@uokufa.edu.iq

Abstract. This research represents the dynamic behavior of the steel tower structural model due to applying seismic load. The 3D finite element model is created using ANSYS-software to simulate the real structure used in experiment. The dynamic properties, natural frequencies and mode shapes, are extracted and compared with the experimental model values. The comparison is implemented to verify the FE analysis procedure of the tower model with experimental model results and they are accepted by 83% tuning ratio. The nonlinear seismic analysis is applied to the structural model using observed ground motion in 1999 Turkey Earthquake. The seismic load is applied in two cases; the first is in the x-direction of the structural tower model and the other in the y-direction of the model in a horizontal plane. The maximum response displacements in x and y-axis and load displacement hysteresis for both loading cases are exhibited. The results display that the tower structural model failed with large response displacement in the first few time of ground motion according to IBC-2018 and NBCC-2005. The collapse of the tower structural model occurred after time 18 sec and the applied seismic loading in x-axis has significant effect.

keywords: seismic loading, modal analysis, steel tower structure, dynamics of structures, earthquake analysis

1. Introduction
In recent years, the earthquakes are more happening in Iraq and in the around region. Therefore, it is important to study the structures that have sensitivity to such type of loading. One of these structures is the tower structure that used for electricity or communication and some scientific papers dealt with dynamic problems of such types of structures.

Yang et al. (2010) carried out dynamic analysis of transmission tower under strong wind load and displayed tower collapse was occurred in some sites, as shown in figure (1) [1]. They used ANSYS-software to simulate and analyze 3D model of cup tower-stay frame. They concluded that this type of structure is important and very sensitive for dynamic loadings and need to increase the stiffness of the structure. Sato & Ishikawa (2012) studied an earthquake resistance of transmission steel towers and comparative analyses of wind resistance data [2]. They built the structural modeling and implemented elasto-plastic analysis of tension square tower with steel pipe members by ABAQUS-software. The tower model has 7471 nodes and 3994 elements with nonlinear beam element for members and mass element for cables. In their nonlinear seismic analysis of structural model, the applied load was a ground
motion at Kobe earthquake in 1995 with maximum acceleration of 617 Gal \(10^2 \text{ m/s}^2\). They concluded that the maximum acceleration at the top node was more 2.5 times than maximum applied acceleration and the residual displacement at top node was 46 mm.

Sharma et al. (2015) implemented comparative analysis of steel telecommunication tower subjected to seismic and wind loading [3]. They created and analyzed the structural modeling of the steel tower structure with different types of bracing using STAAD Pro-software. They carried out modal analysis and response spectrum analyses method for seismic loading. It was concluded that the maximum stress at the bottom members of the tower structure and the natural frequencies were decreased when the height of tower structure increases due to the effect of mass.

In this study, the structural steel tower model is adopted from Lam & Yin (2011) [4]. In addition, the applied ground motion force was recorded in Turkey earthquake in 1999 and selected from pacific earthquake engineering research center (PEER) [5]. The displacement time history is applied on the structural model and the nonlinear transient dynamic analysis is implemented. The maximum displacement response of the tower model is investigated and checked with the seismic displacement requirements. The International Building Code (IBC-2018) specifies the seismic displacement requirements at any node in the structure is not greater than code displacement limit estimated by its equation [6]. Also, the National Building Code of Canada (NBCC-2005) recommended the seismic displacement requirements as the largest drift at any level is 0.025 of the structure height [7].

2. Tower structure description and finite element model

The steel tower structure from experimental study implemented by Lam & Yin (2011) [4], as shown in figure (2-a) is adopted. The experimental 3D structural model was built in the lab of structural vibration at the City University of Hong Kong. The height of the structural model is 2600 mm with eight levels (5-level height of 400 mm and 3-level height of 200 mm). The base level dimensions of the structural model are 500 mm in x-axis and 600 mm in y-axis.
Two material properties were used in the structural model, steel and aluminum, with three types of cross section. The steel circular tube section was used for erecting the columns, while steel circular solid section was used for main beams in the 8-level and the cross arms in the upper of the structural model, as shown in figure (2-a). The horizontal and vertical braces were erected from aluminum hollow square section. The cross sections dimensions and material properties are listed in table 1.

| Material       | Steel         | Aluminum      |
|----------------|---------------|---------------|
| Cross Section Dimension (mm) | Circular Tube | Circular Solid |
|                | $r_{outer} = 6.4$ | $r = 3.2$     |
|                | $t_{tube} = 2$  |               |
|                | Square         | $6 \times 6$  |
| Modulus of Elasticity (N/mm$^2$) | $2.0 \times 10^5$ | $6.8 \times 10^4$ |
| Mass Density (kg/m$^3$)       | $7.8 \times 10^3$ | $2.7 \times 10^3$ |
| Poisson's Ratio            | 0.3            | 0.28           |

In this study, the finite element model is created by ANSYS-software using APDL technique to simulate the experimental structural tower model [8]. The FE model includes 196 elements (160 line elements and 36 Mass elements) and 40 nodes, as shown in figure (2-b). The element types are classified into three groups in the FE model; frame, truss and mass element. The first group of 80-frame elements type BEAM44 represents columns and main beams. The second group of 80-truss element type LINK8...
represents the horizontal and vertical braces in the FE model. The last one is 36-MASS21 element which provided to model the steel plates at joints as 4-part for each 8-level with base supports, as shown in figure (2-b). The mass element is adopted with material properties of \(8\times10^{-3}\) N/(mm/sec\(^2\)) in the z-axis without mass moment of inertia. The four support points in the base of the FE model were assumed fixed.

3. Verification of dynamic analysis procedure
To verify the structural modeling and analysis procedure, modal analysis is carried out on the tower FE model to extract the dynamic properties using Block Lanczos method. The first five natural frequencies values and mode shapes are estimated and compared with the reference [4], as listed in table 2.

| Mode No. | Extracted Freq. from experimental [4] (Hz) | Extracted Freq. from FE model (Hz) | Tuning Ratio (%) | Mode characteristic |
|----------|------------------------------------------|------------------------------------|------------------|---------------------|
| 1        | 37.02                                    | 43.98                             | 81.18            | 1st bending in X direction |
| 2        | 42.99                                    | 50.65                             | 82.16            | 1st bending in Y direction |
| 3        | 68.72                                    | 87.78                             | 72.26            | 1st torsion about Z-axis |
| 4        | 115.86                                   | 105.17                            | 90.77            | 2nd bending in X direction |
| 5        | 127.69                                   | 110.11                            | 86.23            | 2nd bending in Y direction |

From table 2, it is obvious that the average tuning ratio is 83% between the experimental model values in the reference and the computed values from the FE model. While, the mode shapes extracted from FE analysis are identical with the experimental model, as shown in figure (3).
4. Nonlinear seismic analysis of tower structural model

The nonlinear full transient analysis is applied to the FE model using ANSYS-software and used East component of observed ground motion at Duzce in 1999 of Turkey Earthquake as input seismic motion. The acceleration, velocity and displacement time history are shown in figure (4) [5]. The maximum acceleration of the ground motion was 0.114 g and it was produced a maximum displacement of 87.74 mm at 18.59 sec. The magnitude of the earthquake is 7.14 with total period of 42.33 sec recorded with small time steps of 0.01 sec.

![Figure 4](image4.png)

**Figure 4.** Adopted ground motion from Duzce earthquake - Turkey in 1999, M=7.14 [5].

The seismic load is applied in two cases; the first is in the x-direction of the tower model and the second case is in the y-direction. The displacement time history was applied on the FE model and the nonlinear transient dynamic analysis was implemented using Newmark method. The multilinear isotropic hardening material model was adopted to analyze the model using multilinear curve with 280 and 82 MPa yield stress of steel and aluminium material, respectively. The maximum displacement response of the tower model is investigated. The maximum displacement means the initial value of maximum displacement occurred at any node in the FE tower model exceed seismic displacement requirements that equal to 50 mm according to NBCC-2005. For seismic analysis with loading applied in x-direction of the model nodes, the maximum displacement response is 183 mm at node 38 in the top tower reached at time 13.94 sec of the load time history, as shown in figure (5).
Figure 5. Maximum displacement at node 38 of the steel tower model at time 13.94 sec due to ground motion on x-direction.

The maximum displacement response in x and y-axis of node 38 is shown in figure (6). It is clear from figure (6-a & b) that the displacement at the node reached more than 500 mm after time 14 sec in x and y-axis.

Figure 6. Horizontal displacement response of node 38 in structural model due to ground motion in x-direction.
The load-displacement hysteresis of node 38 in x-axis with total shear of base nodes in the structural model due to ground motion x-direction is shown in figure (7). The displacement of node 38 in x-axis is more than 125 mm with about 13000 N total base shear. It is obvious from figure (7) that the structural behavior response in few first second at node 38 in x-axis exceeds the seismic displacement requirements.

![Graph showing load-displacement hysteresis](image)

**Figure 7.** Load-Displacement response of node 38 in x-axis of the structural model due to ground motion x-direction.

After that, the collapse of the tower model occurred in few seconds at time close 18 sec of ground motion history applying in x-direction, as shown in figure (8).

![Image of tower model collapse](image)

**Figure 8.** Failure mode of the steel tower model due to ground motion in x-direction.
For seismic analysis with loading applied in y-direction of the model nodes, the initial value of maximum displacement response is 108 mm at node 38 in the top tower reached at time 17.75 sec of the load time history, as shown in figure (9).

Figure 9. Maximum displacement at node 38 of the steel tower model at time 17.75 sec due to ground motion in y-direction.

The displacement response in x and y-axis of node 38 is shown in figure (10-a & b). It is clear from figure (10-a & b) that the displacement reached more than 500 mm in x and y-axis after time 18 sec.

Figure 10. Horizontal displacement response of node 38 in structural model due to ground motion in y-direction.
The load-displacement hysteresis of node 38 in y-axis with total shear of base nodes of the structural model due to ground motion in y-direction is shown in figure (11). The displacement of node 38 in y-axis is more than 125 mm with about 10000 N total base shear. It is obvious from figure (11) that the structural behavior response in few first second at node 38 in x-axis exceeds the seismic displacement requirements.

![Figure 11](image1.png)

**Figure 11.** Load-Displacement response of node 38 in y-axis of the structural model due to ground motion y-direction.

Then, the collapse of the tower model due to ground motion in y-direction occurred in few time after the recorded time of first loading case in x-direction, as shown in figure (12). That means, the dimension of 600 mm in y-axis of the tower model compared with 500 mm in x-axis increases the stiffness of the structural model. The collapse of tower model produces from set of problems such as material nonlinearity, dynamic nonlinearity and the members failure.

![Figure 12](image2.png)

**Figure 12.** Failure mode of the steel tower model due to ground motion y-direction.
5. Conclusions

This study displays the dynamic behavior of the steel tower structural model under seismic loading. The experimental structural model is simulated and analyzed using ANSYS-software by APDL technique. The dynamic properties, natural frequencies and mode shapes, are extracted for the first five modes. The FE analysis process is verified by comparing the extracted values with the experimental model values in the literature and the tuning was 83%. The nonlinear seismic analysis is implemented for two cases of applied load; the first is in the x-direction of the structural tower model and the other in the y-direction. The maximum displacement response in x and y-axis are investigated and recorded at the top of tower model in the end of two arms at node 38. The initial value of maximum displacement is 183 mm at node 38 in x-axis at time 13.94 sec, while it is 108 mm in y-axis at the same node in time 17.75 sec. The load-displacement hysteresis of tower model explains the seismic behavior of the tower model at node 38 with shear forces in the tower base. The displacement exceed 125 mm with about 13000 N when applying seismic load in x-direction and 10000 N for the loading in y-direction. The results exhibit that the steel tower structural model failed with large response displacement at first few time of ground motion and the applied seismic loading in x-direction of the model is significant effect than in y-direction. The nonlinear transient analysis procedure presented acceptable results and could be used in other structural models in future work. Also, the two-directional seismic loading, collapse problems and real full scale structure can be studied.

References

[1] J. Yang, F. Yang, Q. Li, D. Fu and Z. Zhang, 2010, "Dynamic response analysis and disaster prevention of transmission line under strong wind", International Conference on Power System Technology, Chaina.
[2] Y. Sato and T. Ishikawa, 2012, "Earthquake resistance of transmission steel towers", 15th International Conference of WCEE, Japan.
[3] K. Sharma, S. Duggal, D. Singh and A. Sachan, 2015, "Comparative analysis of steel telecommunication tower subjected to seismic and wind loading", Civil Engineering and Urban Planning: An International Journal, vol.(2), No.(3), pp. (15-33).
[4] Heung-Fai Lam and Tao Yin, 2011, "Dynamic reduction-based structural damage detection of transmission tower: practical issues and experimental verification", Engineering Structures, vol. (33), pp.1459-1478.
[5] https://peer.berkeley.edu.
[6] International Building Code (IBC), 2000, International Code Council, Fall Church, Virginia, USA.
[7] National Building Code of Canada (NBCC), 2005, National Code, Ottawa, Canada,
[8] Ansys software, Release 18, version 18-1.2 Inc, 2016.
[9] Shubham Sharma and Ahmed J. Obaid 2020 J. Phys.: Conf. Ser. 1530 012124.