*International Journal of Engineering & Technology*, 7 (4.20) (2018) 491-495

SPC

**International Journal of Engineering & Technology**

*Website: www.sciencepubco.com/index.php/IJET*

***Research paper***



# Numerical Evaluation of Seismic Response of Asymmetrical Reinforced Concrete Frame Buildings

**Mohammed J. Hamood[1], Layla A. Ghalib[2], Ameer G. Abdalwahab[3]**

*Civil Engineering Dept., University of Technology / Baghdad-Iraq[1,2,3]*
**Corresponding author E-mail: mjh_smm@yahoo.com[1]*

**Abstract**

Asymmetrical multi-storey buildings are almost unavoidable in modern structures due to various types of useful and architectural requirements. Latest earthquakes showed that irregular distribution of mass, stiffness and strength cause serious damage in building structural systems. This paper investigates the numerical simulation of buildings with plan irregularity and presents a case study to demonstrate the numerical evaluation of the seismic response of a three real plan-asymmetric reinforced concrete building tested at full scale at the European Laboratory for Structural Assessment of the Joint Research Center, Ispra / Italy within the SPEAR project. The structural evaluation performed through a validated Finite Elements Package, modeled by the general purpose ABAQUS, which is able to run accurate analysis, in particular nonlinear static and dynamic analysis considering both geometric nonlinearity and material inelasticity.
Adequacy of the numerical modeling is verified by comparing numerical and experimental results through evaluation of the seismic capacity and dynamic characteristics of the building. The provisions of the adopted seismic code for designing such buildings are also checked over and done with the nonlinear static and dynamic analysis by verifying the proficiency of an analytical model for simulating the nonlinear response of structures considered to conduct an investigation into experiments.

***Keywords***: *seismic response; asymmetrical frame buildings; finite element analysis; ABAQUS.*

## 1. Introduction

Asymmetry in plan causes torsional unbalanced in a building because the centers of mass and of rigidity do not coincide, which may create large displacement extensions and high force concentricity within the resisting elements that may proceed serious damages. Symmetric and regular structure that is properly designed has a much higher capability to resist a strong earthquake occurrence than asymmetric structure and its response to earthquake loading is far more straightforward to predict and design for. However, most buildings have a number of irregularities in the geometric layout or the distribution of mass, stiffness, and/or strength. As it is known, a structure that is irregular has non-preferable seismic perform, which involves having specific part of structure with a high plastic concentricity. Magliulo et al., 2014 [1] conducted a study to examine the seismic behavior of regular and irregular structures. They chose three typical Italian reinforced concrete buildings, using different nonlinear analytical methods. Results revealed that the incidence angle of the seismic input motion has a great impact on the performance of plan-asymmetric reinforced concrete structures. All recent building codes provide guidelines to confirm the regularity of structures either in plan or in elevation; if the rules are not achieved, some "penalties" in the design are providing (CEN, 2004) [2].
Aval S. B. B., Asayesh M. J., 2017 [3] investigated the seismic behavior of two, five, and fifteen storey asymmetrical reinforced concrete building Tunnel Form buildings with two distinct plans. They found an excellent seismic response of this form of structure in spite of its specific irregularity. The high lateral stiffness and strength of buildings compensate for the comparatively low torsional rigidity with respect to lateral stiffness. Accordingly, the building showed superior seismic execution in high seismic activity.
The use of computer programs to achieve more complex numerical models for dynamically analysis structures has improved significantly in the preceding years. In order to have confidence in the outputs of these complex analyses, it has become increasingly important to verify and validate the software against literature case studies and experimental data. Seismic behavior of irregularly plan structures may be assessed using time history analysis, but, this method is, in common, prolonged and has to be repetitive several times to have a varied set of consequences that could describe the response of the structure excited by a seismic loading. Consequently, by knowing how the building will respond to a dynamic force of particular frequencies, we can modify and develop the design of a building in a region where earth vibrations are frequent. Furthermore, studying dynamic response can simply save the building from collapsing under dynamic loads, and proper understanding of the seismic behavior of irregular building needs to be studied in details before designing.

## 2. General description of the test building

Indeed, this structure has been modeled by other researchers in the past who have however employed different software solutions for numerical modeling. To predict the large displacement behavior of three dimensional building when loading statically or dynamically, the product ABAQUS was utilized, in this. The examined building depicts the displaying of a full-scale three-storey fortified solid casing working, according to the 1954-1995 Greek Code. The

**Copyright © 2018 Authors. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.**

structure has been built with the development practice and materials utilized in Greece in the mid 70's, the auxiliary framework uncovers a few imperfections when considering the principle standards of seismic tremor safe plan. It is customary in tallness yet sporadic in plan. The model building has been tried at (ELSA) of the Joint Exploration Focal point of Ispra/Italy under pseudo-dynamic condition utilizing the Herceg-Novi bi-directional accelerogram enlisted all through the Montenegro 1979 seismic tremor.

The building was intended to withstand just gravity stacks in a way that is unique in relation to past ordinary and seismic structure codes-based structures. Thus, this building experiences basic lacks when reacting to seismic tremor loadings. Another frail point that describes joints of the building is neither pillar nor section stirrups spread into them. That is; there is no repression by any stretch of the imagination. Additionally, a few pillars specifically meet different bars making as bar to-shaft joints that have no segment bolster. Plain fortification steel bar was utilized that is portrayed in the present creation by being rare. The diagram of the building together with the arrangement of a common tedious floor is displayed in Figure (1). Essentially, basic specialists ignored the impact of stone work boards while choosing the auxiliary framework; this was on the grounds that they consider these block workmanship boards are flimsy when contrasted with the edge. That was the reason; the examination neglected infill dividers and stairs. What made the segments more grounded along the Y course, the most grounded bearing of the building, was that segment C6 had a cross-segment of 250 by 750 mm, though the rest segments had a square 250 by 250 mm cross-segment. For more insights about the components of the test building and part support are all displayed in Figures (2 and 3). Additionally, piece thickness was 150 mm while add up to pillar profundity was 500 mm. The basic plan of establishment was performed in a way that bolsters test outline working to be steadily based on unbending floor of the lab. That is; all establishment hubs are completely limited against any revolution and interpretation. For more data about the test allude to Fardis (2002) [5].

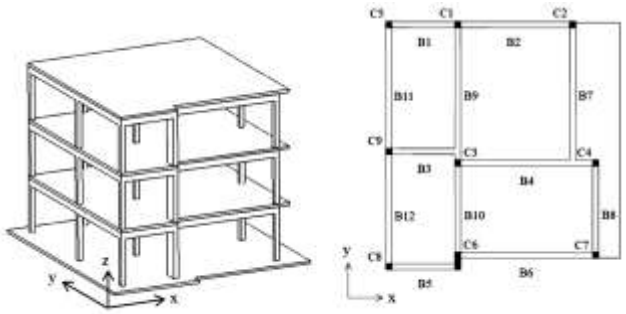



**Fig. 1:** Three storey test building configuration [4]

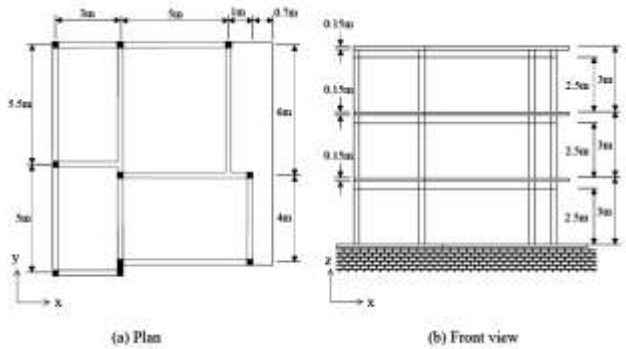



**Fig. 2:** Outline and dimensions of the test building [4]

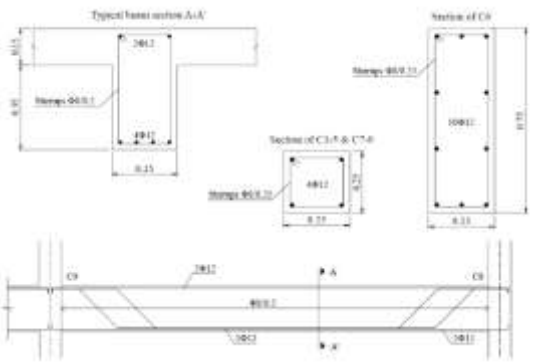


**Fig.** 3: Typical member cross sections details (dimensions in mm) [4]

## 3. Materials properties

The bare frame buildings were modeled with three-dimensional finite elements models by using numerical analysis. Considering the non-linear material behavior when predicting reinforced concrete members' response can be met through accurately modeling the uniaxial material stress-strain behavior. Thus, the results of the lab test with values in Table (1) and Figure (4 (a, b)) were used in the material properties of steel and stress-strain relationship as demonstrated by Jeong S.H. and Elnashai A. S. (2004) [4]. Where for steel, the average yield strength of 360 MPa, ultimate strength of 450 MPa and Young's modulus of 206000 MPa was assumed. As the concrete compressive strength of concrete (f'c), it was 25 MPa while the constitutive relationship suggested by Mander et al. (1988) [6] was adopted to achieve two purposes; First, adjusting the behavior of unconfined concrete, and second predicting the confining effect K; i.e., the ratio of the confined concrete strength (f'cc) to the plain concrete strength (f'c). The lack of a sufficient transverse reinforcement has led the confinement effect not to be considered when calculating the core concrete and the confinements factor K and considering it to be close to 1 for all members. Consequently, the factor was roughly estimated to be 1.01 in the analytical model.

**Table 1:** Rebar properties according to material test results from ELSA [4]

| Bar Φ (mm) | Yield strength (MPa) | Ultimate strength (MPa) | Yield strain | Ultimate strain | Young's modulus |
|---|---|---|---|---|---|
| 8 | 467 | 583.67 | 0.00227 | 0.131 | 206000 |
| 12 | 458.67 | 570.33 | 0.00223 | 0.174 | 206000 |
| 20 | 376.67 | 567.33 | 0.00183 | 0.168 | 206000 |

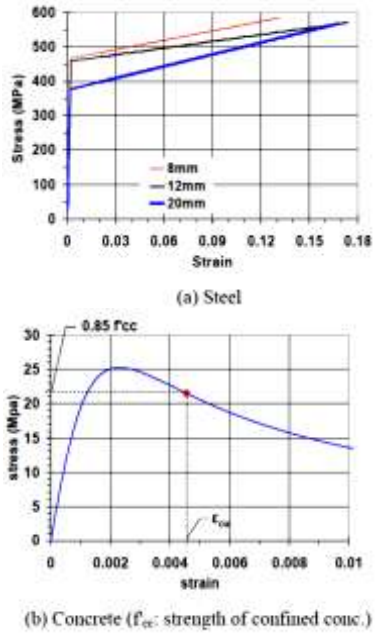



**Fig. 4:** Stress-strain relationships of materials used [4]

## 4. Gravity loading determination for testing

To apply the planned gravity burdens to the thought about building, water tanks were utilized as appeared in Figure (5 and 6). The focal point of weights of these tanks was put at the focal point of weights of pieces; such a stage helps give a comparable hub constrain connected on sections as the uniform load conveyance [4]. Talking about the scientific model the gravity loads was estimated by figuring parts of the planned gravity stacks that exist on the chunk and oneself weight of the structure itself. Along these lines, add up to dead load and 30% of live load were considered for the gravity stacks in the examination. As needs be, they were set up to be (0.5 kN/m2) and (2 kN/m2) for both for completing live loads individually.



**Fig. 5:** Water tank distributions [4]

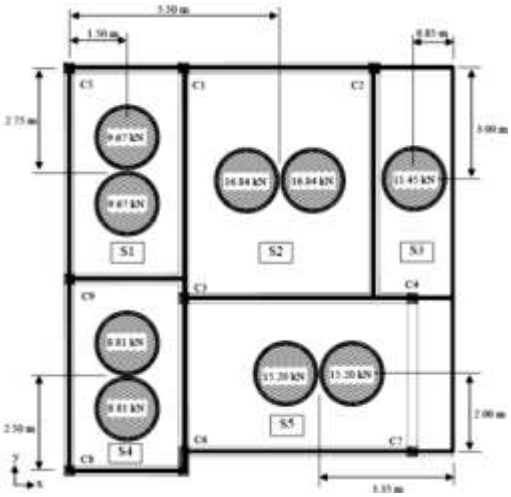



**Fig. 6:** water tanks distributions on each slab [4]

## 5. Determination of actuator movement

A full scale test was improved the situation the model at Ispra inside the Lance venture. For more data about the structure and its pseudo-powerful testing might be acquired from (Mahin and Shing, 1985, Negro, 1996 and Molina et al., 1999) [9, 10, and 11] see Figure (7). The Lance building is lopsidedly planed in both X and Y bearings; in any case, its height is ordinary. Actuator developments were anticipated utilizing numerical reenactments. Such desire distinguishes whether the inadequacies of stacking cylinders were sufficient for the testing or not. Figure (8) outlines the obviously places of the actuators and the development of every single one of them.

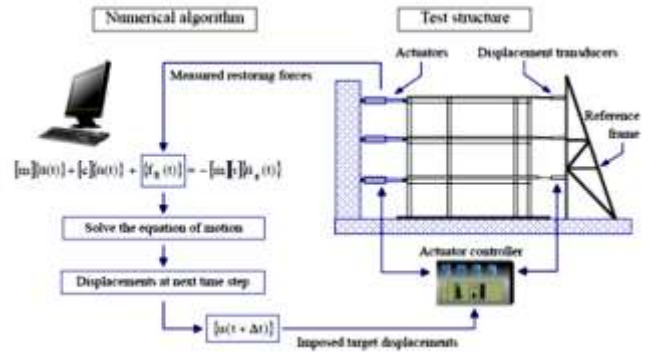



**Fig. 7:** Graphic representation of pseudo-dynamic test [4]

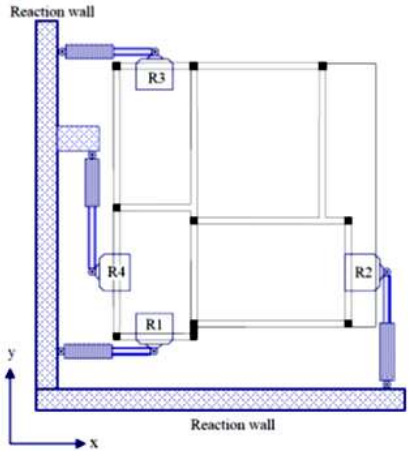
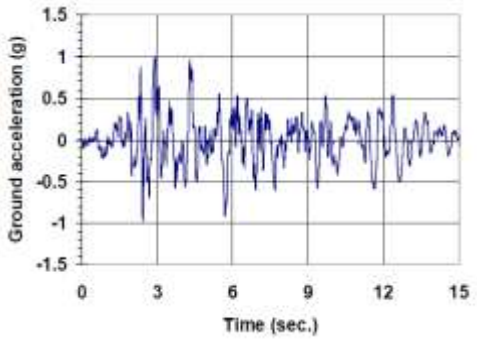
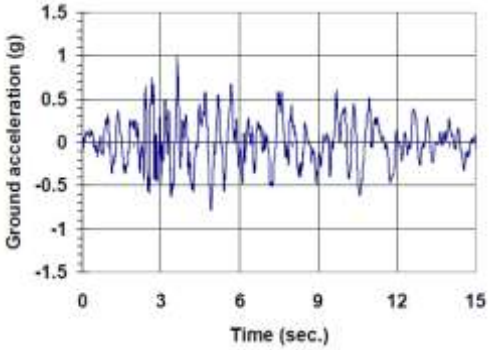



**Fig. 8:** Description of actuators, Position and direction of actuators [4]

The SPEAR building was examined pseudo-dynamically using two directional loadings based on a ground motion recorded at Herceg Novi station throughout the 1979 Montenegro earthquake Figure (9 (a, b)) and scaled to match with the EC8 type-I spectrum for soil type (C). A bi-directional record was applied to the structure during three runs of the linearly increasing intensity of the peak ground acceleration (PGA), such as 0.12 g, 0.15 g and 0.20 g.




(a) In the X-direction

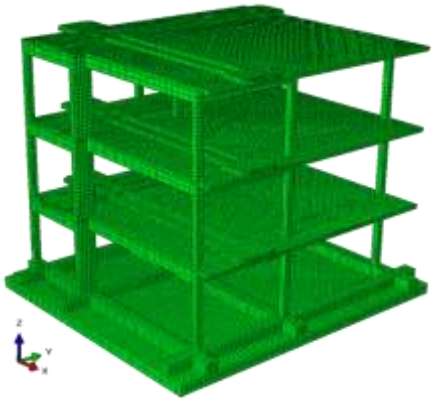



(b) In the Y-direction

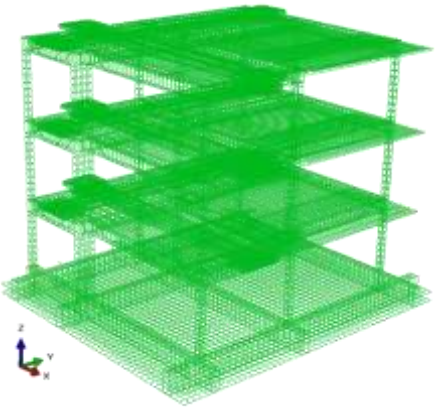
**Fig. 9:** Acceleration response history of Montenegro 1979 - Herceg Novi [4]

## 6. Verification of the problem simulation

Model verification is critical in the development of a simulation model. The computer program ABAQUS/CAE 6.14 [12] used in this study concerning the effect of dynamic vibration, the results that have been obtained from ABAQUS program were compared with the results of an experimental results of a full scale test. The time increment for the time history analysis has been set as automatic in options of step process for dynamic explicit analysis, which is corresponding to the input record selection time step.



**(a)** Finite Element Model for concrete



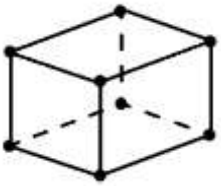
**(b)** Finite Element Model for steel reinforcement

**Fig. 10:** Simulation of reinforced concrete structure (at full scale) in ABAQUS program

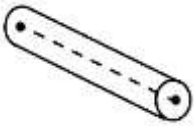
Three dimensional first order reduced integration continuum elements (C3D8R/8-node Brick) are used in modeling the concrete members whereas the steel rebar are modeled by utilizing (T3D2/2-node linear 3D Truss) elements. These elements are versatile and may be used in modeling the simple linear analysis or the complex nonlinear analyses relating inelastic and large deflections. The Concrete Damaged Plasticity Model (CDPM) has been used for concrete modeling.

The frame building, and supports were modeled by creating parts of beams, columns, footings and reinforcements in ABAQUS. Then elements generated through nodes with auto-numbering of elements. The finite element model of the reinforced concrete building with typical mesh discretization of the concrete and steel rebar is shown in Figure (11 (a, b)). The typical solid elements in ABAQUS are shown in Figure (12 (a, b)).

To attain valid results from the (C3D8) elements, the using of a rectangular mesh is recommended. So, the mesh was set up and square or rectangle elements were generated. The width and length of elements should be consistence with the elements and nodes for concrete parts in the modeling.



(a) Concrete Discretized



(b) Steel Reinforcement Discretized

**Fig. 12:** Solid Element used (Abaqus/CAE 6.12 Analysis User's Manual 2012) [12].

## 7. Comparison of experimental results and ABAQUS program results

The same input motion was used to verify the adequacy of the current analytical model. Table (2) and figures (13) to (16) show the comparison for Top Displacement and base shear between Experimental results in ELSA and ABAQUS program results with good agreement, so that the same model can be used for further studies with confidence. As observed, the numerical model is slightly stiffer that the test specimen, as was expected. Commonly, designing and analyzing reinforced concrete structures are performed as in the form of a bare frame without taking into account the impact of the infill segments to both strength and stiffness. Despite the fact that during earthquakes, the infill walls adjust the performance of the building, such a change diffrs from that expected for bare frame structures.

**Table 2:** Difference in Top Displacement by ABAQUS vs. ELSA test

| Type | Direction | Max Top Displacement (mm) | Max Base Shear (KN) | Percentage Difference for Top Displacement | Percentage Difference for Base Shear |
|---|---|---|---|---|---|
| (ELSA) Experimental results | X | 70.12 | 173.00 | | |
| | Y | 47.43 | 256.82 | | |
| ABAQUS program results | X | 83.81 | 244.13 | 16.3 % | 29.1 % |
| | Y | 54.12 | 322.19 | 12.4 % | 20.3 % |

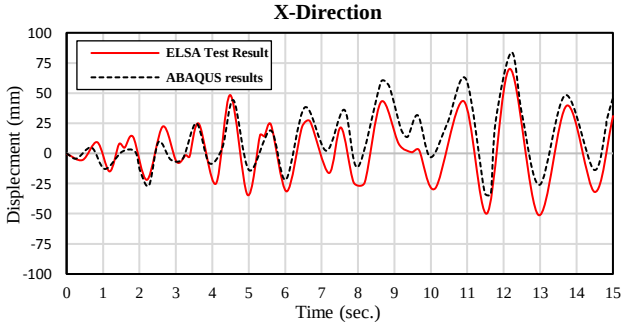



**Fig. 13:** Experimental vs. Numerical results of the Top Displacement Histories of ELSA at center of mass, 0.15g PGA testing and that obtained from ABAQUS (in X-direction)

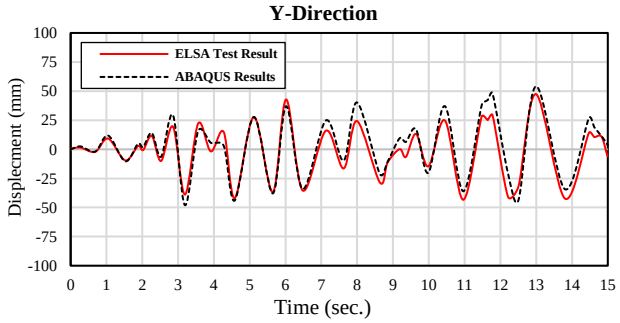



**Fig. 14:** Experimental vs. Numerical results of the Top Displacement Histories of ELSA at center of mass, 0.15g PGA testing and that obtained from ABAQUS (in Y-direction)

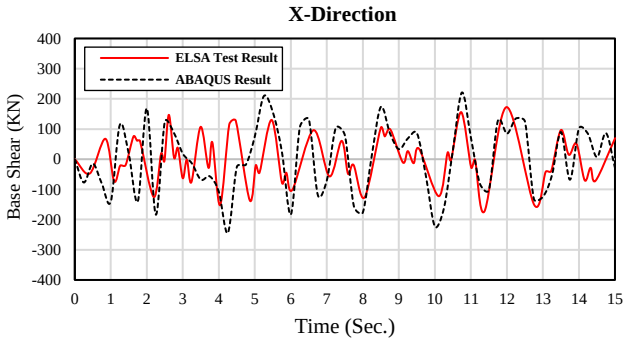



**Fig. 15:** Comparison of Base Shear from experimental results and Numerical results in X-direction

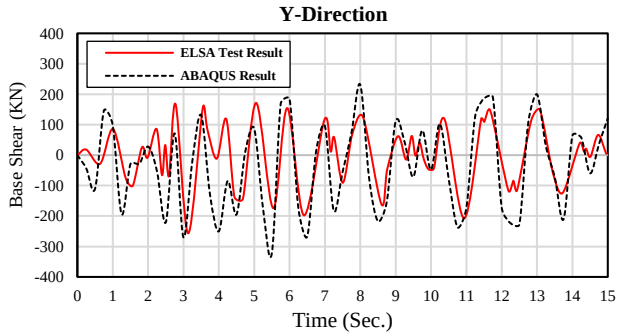



**Fig. 16:** Comparison of Base Shear from experimental results and Numerical results in Y-direction

## 8. Conclusion

Based on the results presented above, it may be concluded that the nonlinear time-history analyses were able to predict comparatively well the behavior of the building structure, mainly in terms of storey displacements. So, further investigations considering different parameters are clearly needed.

For wholeness, the experimental results of the original building structure have been compared in terms of top displacement with the numerical simulations. Concluded simulation reinforced concrete structure in ABAQUS program may yield a responsible percentage difference of Top Displacement in X-Direction and Y-Direction between Experimental results in ELSA and ABAQUS program results of 16.3% and 12.4% in X and Y-Direction respectively. also for Base Shear 29.1% and 20.3%. The numerical modeling together with the simulations helps predict the most proper input motion history and intensity. Such prediction helps investigate the performance and seismic response of the intended structure.

## References

[1] Magliulo G., Maddaloni G. and Petrone C., "Influence of earthquake direction on the seismic response of irregular plan RC frame buildings", Earthquake Engineering and Engineering Vibration, Vol.13, Issue 2, (2014), pp:243–256, https://link.springer.com/article/10.1007/s11803-014-0227-z

[2] CEN (2004), Eurocode 8, Design of Structures for Earthquake Resistance-Part 1, General Rules, Seismic Actions and Rules for Buildings, EN 1998-1, Brussels, Belgium.

[3] Aval S. B. B., Asayesh M. J., "Seismic performance evaluation of asymmetric reinforced concrete tunnel form buildings" The Institution of Structural Engineers, Volume 10, (2017), Pages 157–169, https://www.sciencedirect.com/science/article/pii/S2352012417300176

[4] Jeong S. H., Elnashai A. S., "Analytical Assessment of an Irregular RC Full Scale 3D Test Structure", research is supported by the Mid-America Earthquake Center under National Science Foundation Grant EEC-9701785, (2004), pp:141.

[5] Fardis M.N., "Design of an Irregular Building for the SPEAR Project-Description of the 3-Storey Structure", Research Report, (2002), University of Patras, Greece.

[6] Mander J.B., Priestley M.J.N. and Park R., "Theoretical stress-strain model for confined concrete", ASCE Journal of Structural Engineering, Vol. 114, Issue 8, (1988), pp. 1804-1826. https://ascelibrary.org/doi/10.1061/%28ASCE%290733-9445%281988%29114%3A8%281804%29

[7] Takanashi K., Udagawa K. and Tanaka H., "A simulation of earthquake response of steel buildings," Proc. of the 6th World Conference on Earthquake Engineering, New Delhi, Vol. 3, (1977), pp:3156-3162.

[8] Takanashi K., Udagavw K. and Tanaka H., "Pseudo-dynamic tests on a 2-storey steel frame by computer-load test apparatus hybrid system," Proc. of the 7th World conference on Earthquake Engineering, Istanbul, Turkey, Vol. 7, (1980), pp:225-232.

[9] Mahin S.A., Shing P.B., "Pseudo-dynamic method for seismic testing", ASCE Journal of Structural Engineering, Vol. 111, Issue 7, (1985), pp: 1482-1503. https://ascelibrary.org/doi/10.1061/%28ASCE%290733-9445%281985%29111%3A7%281482%29

[10] Negro P., Pinto A.V., Verzeletti G. and Magonette G.E., "PsD test on four-story R/C building designed according to Eurocodes", ASCE Journal of Structural Engineering, Vol. 122, Issue 12, (1996), pp: 1409-1417, https://ascelibrary.org/doi/10.1061/%28ASCE%290733-9445%281996%29122%3A12%281409%29

[11] Molina F.J., Verzeletti G., Magonette G., Buchet Ph. and Geradin M., "Bi-directional pseudodynamic test of a full-size three-storey building", Earthquake Engineering and Structural Dynamics, Vol. 28, Issue 12, (1999), pp: 1541-1566, https://onlinelibrary.wiley.com/doi/10.1002/%28SICI%291096-9845%28199912%2928%3A12%3C1541%3A%3AAID-EQE880%3E3.0.CO%3B2-R

[12] Abaqus/CAE 6.12-1 Analysis User's Manual 2012. Dassault Systèmes Simulia Corp., Providence, RI, USA.