Journal of Simulation and Analysis of Novel Technologies in Mechanical Engineering
13 (2) (2021) 0039~0046
HTTP://JSME.IAUKHSH.AC.IR

Assessing the vulnerability of Shiekh Bahaei's bath under the blast wave

Melika Honary[1], Abdullah Vaez Shoushtari[2*], Erfan Jaberzadeh[1]

[1]*Department of Civil Engineering, Khomeinishahr Branch, Islamic Azad University, Khomeinishahr/ Isfahan, Iran*
[2]*Department of Civil Engineering, Isfahan (Khorasgan) Branch, Islamic Azad University, Khorasgan/ Isfahan, Iran*

[*]*vsabdollah@gmail.com*

(Manuscript Received --- 24 Jan. 2022; Revised --- 14 Feb. 2022; Accepted --- 15 Feb. 2022)

Abstract

The purpose of this study is to investigate the vulnerability of Shiekh Bahaei's bath under the blast wave. This study was performed as a numerical study using software analysis. The method of three-dimensional nonlinear finite element analysis was selected by Abaqus software. Nonlinear dynamic analysis method was selected to determine the moment-to-moment response of the structure to the explosive load. The Drucker Prager criterion is also used to model building materials in Abaqus software. The accuracy of finite element modeling results was ensured by comparing the modeling results with the laboratory sample. The finite element model of Shiekh Bahaei's bath was modeled in Abaqus software and nonlinear dynamic analyses under the blast wave were performed. Then, the results were compared in the form of tensile damage meters and displacement-time diagrams. Based on the results of this study, the location of the breakdown and the amount of displacements in different parts of the bath under the blast wave were determined.

Keywords: Shiekh Bahaei's bath; Vulnerability; Explosion wave; Nonlinear dynamic analysis.

1- Introduction

The study of passive defense mechanisms is a fundamental issue in today's asymmetric wars to counter hostile aggression and reduce the damage caused by air and naval attacks. In accord with the opportunity provided in peacetime, it is necessary to arrange some plans. Iran has been always exposed by foreign threats due to its privileged strategic position and rich natural resources. It is necessary for structural and earthquake engineers to know the threats, and how to deal with them. Many existing structures are vulnerable to loads due to blast waves and therefore their resistance to such loads should be increased [1].

An explosion is the very rapid release of energy in the form of light, heat, sound, and a shock wave. A shock wave consists of very compressed air that propagates a wave radially at supersonic speeds from the surface of the explosive to its surroundings. Damage caused by the shock wave can be both direct effects due to an explosion and effects because of a progressive failure.

For a long time the subject of the bath has been considered by scholars, jurists, physicians, anthropologists, architects and tourists from various perspectives due to its role in health, religious status, medicine, architecture and its importance. Baths in the ancient urban context have played a key role in demographic centralism, news exchange, friendship building, planning for social, political, economic and religious affairs and a place for cultural exchange of different sections of society. Texts and sayings state that this bath had a treasury whose water was heated automatically and without consuming energy, and therefore its construction is attributed to Sheikh Bahaei, Bahr al-Ulum of his time, and this is why it is world famous.

Much research has been done on the seismic performance of masonry structures; however, studies on the effect of blast wave on the performance of masonry structures are limited. In experiments and finite element simulations, Elamps et al. [2] investigated the response of building structures made of brick masonry under horizontal loading. In this research, in addition to laboratory testing, the three-dimensional nonlinear finite element method and Abaqus software were used to model and analyze the results. The analysis performed in this study was nonlinear chronological history. The results of this study showed that the most common failure mechanism that occurs is due to cracking, which is caused by improper connection between the brick and mortar. Damage onset can also occur under the influence of increased stress in the opening corners and in the vicinity of wooden elements. In their research, Valenti and Milani [3] evaluated eight historic towers made of building materials (in northeastern Italy) under nonlinear dynamic and static analysis. In this research, 3D nonlinear finite element method and Abaqus software were used to model and analyze the results. The results of this study indicate the high vulnerability of historic towers with building materials. In addition, the results obtained from nonlinear static analysis have a good agreement with the results of nonlinear dynamic analysis. Tiberti et al. [4] investigated the causes of damage and hypotheses to reduce seismic damage. The results of this study showed that insufficient strength of the constituent materials is the main cause of damage to the structure. Partial improvement of a part of the wall by the municipality can be useful in reducing the damage occurred in successive earthquakes for the structure. Valenti et al. [5] seismically evaluated two Baroque churches made of building materials (in Italy) damaged in 2012 Emilia earthquake. In this research, 3D nonlinear finite element method and Abaqus software were used to model and analyze the results. Numerical simulations in this study led to the determination of damage distribution, identification of the most vulnerable structural elements and highlighting the main defects of church structures when exposed to different seismic levels. Hejazi et al. [6] in 2016, in their research to optimize the shape of historic Iranian brick domes with different geometric shapes and under uniform pressure, determine the failure load under concentrated load and buckling load under uniform pressure of domes with optimal cross section they paid. The studied domes have a semicircular cross section, a goat horn and five sevens. The failure load under the concentrated load of optimally shaped domes is higher than non-optimally shaped domes. The buckling load of optimally shaped domes under uniform pressure is much higher than the failure load of them under the same load. The buckling load increases with decreasing span. The highest and lowest buckling loads are related to goat horn domes and semicircular domes, respectively. In 2013, Akhvisi et al. [7] in their research analyzed the failure of unreinforced masonry structures based on stiffening and strain softening in the framework of a multi-plane model. Comparison of numerical predictions of nonlinear analysis of unreinforced masonry structures against lateral loads with the results obtained from experimental data shows the ability of this model to analyze the failure of masonry structures. Tai and Kim [8] analyzed the performance of reinforced concrete members under laboratory and numerical analysis under the blast wave. In this research, six sets of reinforced concrete columns with different dimensions under the blast wave with different intensities were investigated. The results of this study showed that short and thick columns under the blast wave only have local buckling. But thin columns under the blast wave, in addition to local buckling, also suffer from total failure. Bozorgvar and Shoushtari [9], in 1390, investigated the effects of explosion on earthquake-resistant concrete structures. In this study, the behavior of earthquake-resistant concrete structures designed according to Regulation 2800 against explosive loads was investigated. For this purpose, a 4-storey concrete residential building was subjected to different loading compositions and was analyzed in 3D using ABAQUS finite element software. Then the performance of the structure under these loadings was investigated. Then, the responses due to explosion loading, such as base shear and floor displacement, were compared with seismic responses, and the comparison of the responses resulting from explosion and earthquake showed that the duration of explosive loads, although very It is less than an earthquake, but it creates a base cut and more displacement in the building. According to the results, earthquake-resistant structures should be re-evaluated to withstand explosive loads. Khandlwal et al. [10] investigated the behavior of braced steel structures due to explosion and progressive failure. In this study, 10-story steel structures with coaxial bracing system and eccentric bracing system were studied in two dimensions. The results showed that when the braces are placed around the structure, the structures with the eccentric bracing system show better behavior than the coaxial bracing system. Abroshan and Moradloo [11] investigated the geometric nonlinear behavior of composite shear walls under explosive loading. The results show that for the intensity of the loads considered in this analysis, the calculation of geometric nonlinear behavior will reduce up to 14% in the peak displacement and increase by 3.5% the main stress and strain in the center of the wall. It is expected that the combination of geometric nonlinear behavior with material nonlinear behavior of these structures will lead to a better understanding of the behavior of the structure. Schallen et al. [12] investigated the response of building structures under the blast wave. In this research, Ansys software was used. Explosive loads were applied to the structure at two different locations and at different distances. The results of this study showed that the effect of the blast wave decreases with increasing wave distance from the structure. In this research, the finite element model of Shiekh Bahaei's bath is modeled in Abaqus software and analyzed under the blast wave, and the results are compared in the form of tensile damage meters and displacement-time diagrams.

2- Modeling and validation of laboratory sample results by FEM

Before modeling finite elements, to study its behavior, one must ensure the accuracy of the modeling method and the results predicted by the finite element method. To do this, the results measured in a valid experiment should be compared with the values obtained from the finite element model corresponding to the tested sample, and if the results are consistent, the results of software modeling can be trusted. In this section, the laboratory model developed by Tai and Kim [8] is introduced as a connection reference sample for use in validating the results of primary finite element modeling. In this research, a square reinforced concrete member with a cross section of 230 × 230 mm and a length of 2440 mm was tested. According to Fig. 1, the end of the reinforced concrete member is trapped using semi-buried blocks. Boundary conditions and the applied loads to the concrete member are shown in Fig. 2. As shown in Fig. 2, an eight kg equivalent of TNT is placed at 320 mm from the end of the base at a height of 600 mm. The explosion charge curve was applied to the column according to Fig 3. The specifications of the materials used are presented in Tables 1 and 2. For concrete modeling by Abaqus software, the concrete damage plasticity material available in the predefined material library of this software was used. The compressive and tensile strength of concrete is 30 MPa and 3.5 MPa, respectively.

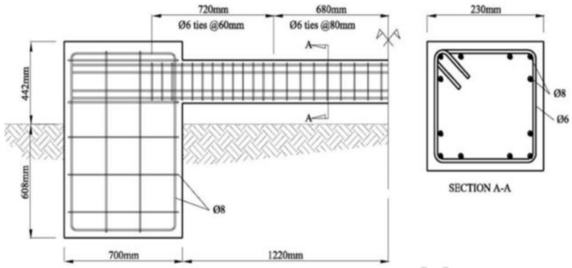


Fig. 1 General shape and specifications of laboratory sample [8]

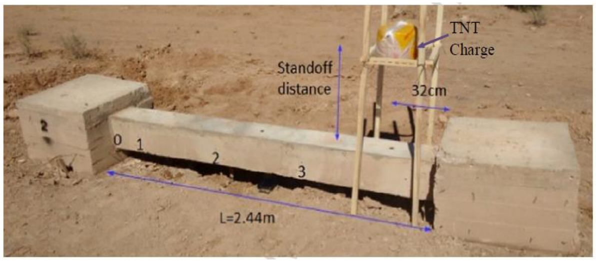


Fig. 2 Boundary conditions and loads applied to the concrete member [8]

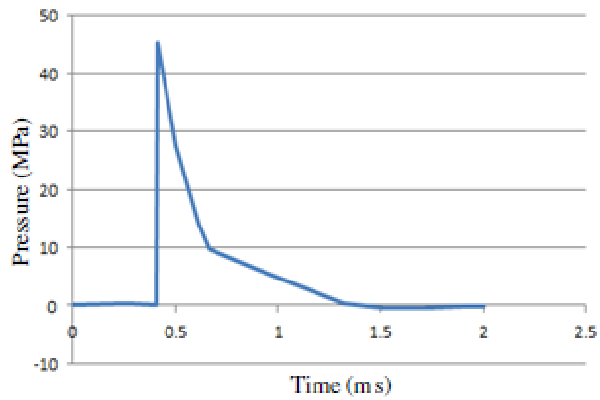


Fig. 3 Load applied to the column[8]

Table 1: Mechanical properties of concrete [8]

Specific Mass (kg/m^3)	Elasticity modulus (GPa)	Poisson's ratio	Compressive strength of concrete (MPa)
2400	26.41	0.2	30

Table 2: Mechanical specifications of elastoplastic model for steel [8]

Yield stress (MPa)	Poisson's ratio	Modulus of elasticity (MPa)
420	0.3	205000

The process of assembling the model in the software is shown in Fig. 4. Loading and boundary conditions are applied to the reinforced concrete member according to Fig. 5. According to this figure, the bottom of both bases is fastened with a support and the blast load is applied to the column near the left base. For modeling concrete by ABAQUS software, the concrete damage plasticity module available in the software library of the predefined materials of this software is used. The plastic damage model can show the crack distribution trend at each stage of loading. Cracks always appear on the plane perpendicular to the original plastic stresses. Cracks appear in the structure when the maximum stresses at the point of integration of the solid element exceed the final tensile strength of the material. The appropriate yield criterion for steel materials is von Mises criterion which was used to model steel in software. In this research, the Solid element has been used to model the concrete member. The C3D8R element was selected to model the concrete part. This element is a three-element, eight-node element that uses the reduced integration method to solve integrals. The T3D2 element is also used to model longitudinal and transverse bars, which is a two-node three-dimensional element. (Fig. 6).

In order to compare the correlation between the tensile damage distribution and the failure occurred in the laboratory sample and the results of finite element analysis in Abaqus software, Fig. 7 is shown. A comparison of the maximum deformations occurring in the laboratory sample and the finite element model is presented in Table 3.

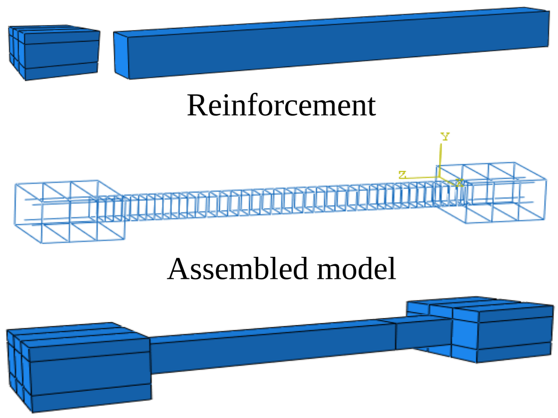


Fig. 4 Model assembly process

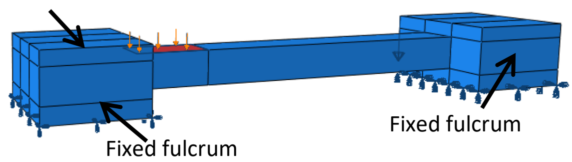


Fig. 5 Construction of boundary conditions and loading for reinforced concrete member

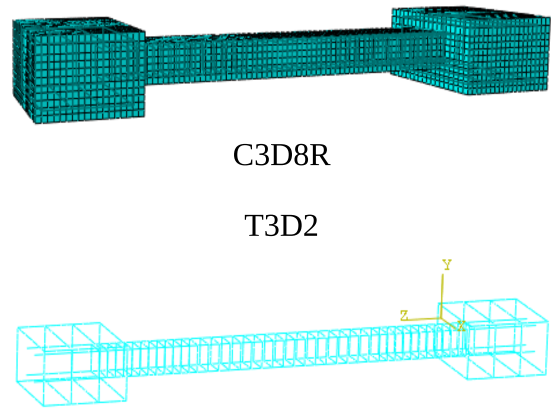


Fig. 6 Model comparison

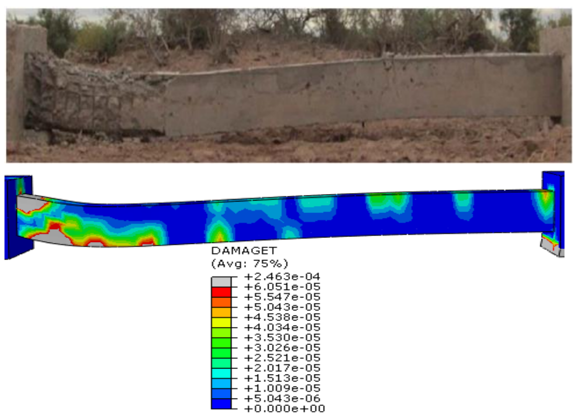


Fig. 7 Correlation of the results related to the distribution of tensile damage and failure occurred in the laboratory sample and finite element model

Table 3: Comparison of the maximum deformations that occurred in the members

Laboratory sample (mm)	FE model (mm)	Percentage difference (%)
67.2	68.5	1.9

3. Modeling Shiekh Bahaei's bath in Abaqus software

Due to the large number of bathroom components and their complex geometric shape; Direct modeling of the bathroom was not possible in Abaqus software; Therefore, first the three-dimensional model of the bathroom was made in AutoCAD software and then this model was entered in Abaqus software environment. The bath was modeled according to Fig. 8 in Abaqus software. The shape of the elements was chosen as Solid and the type of elements as C3D8R; This element is a three-dimensional, eight-node element that uses the reduced integral method to solve integrals. The mechanical properties of the materials used were defined according to Table 4 and the stress-strain curve according to Fig. 9. The Williams-Warnack failure criterion was used for modeling.

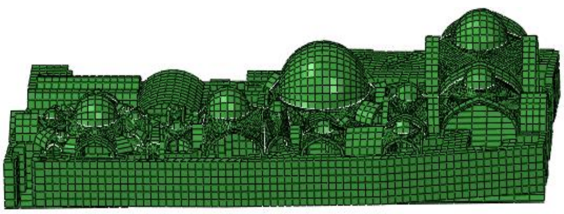
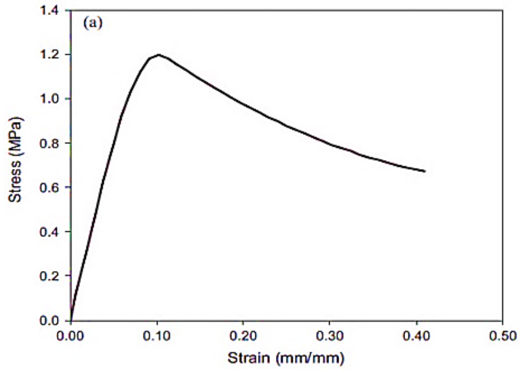


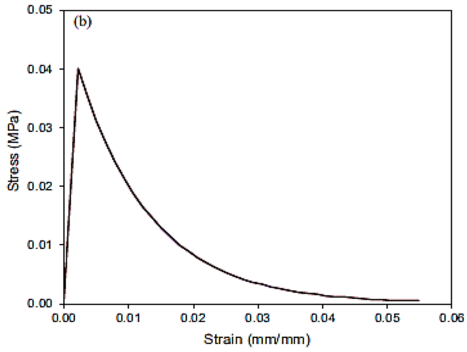
Fig. 8 Model made of bathroom in Abaqus environment

Table 4: Mechanical specifications intended for brick materials

Specific mass (kg/m^3)	elasticity Modulus (Mpa)	Poisson's ratio	Compressive strength (Mpa)	Final strain
1300	18	0.3	1.2	0.1



(a)



(b)

Fig. 9 Stress-strain curve of brick materials: (a) stress-compressive strain, (b) tensile-strain stress [12]

Content-based BlastGM software was used to estimate the amount of shock caused by the explosion on the instrument. This software calculates the amount of acceleration applied to the structure according to the TNT mass value, the distance of the explosion site to the structure and the explosion duration of the diagram (Fig. 10).

3-1- Review of the results

In Abaqus software, concrete plastic damage model can show the trend of crack distribution at each stage of loading. Cracks always appear on the plane perpendicular to the original plastic stresses. Cracks appear in the concrete when the maximum stresses at the point of integration of the solid element exceed the final tensile strength of the concrete. Fig (11) shows the distribution trend of tensile damage, as shown in these figures. According to the results of the research, the most damage occurred at the location of the domes in the southern part of Sheikh Bahaei's Bath. These locations have the highest altitude.

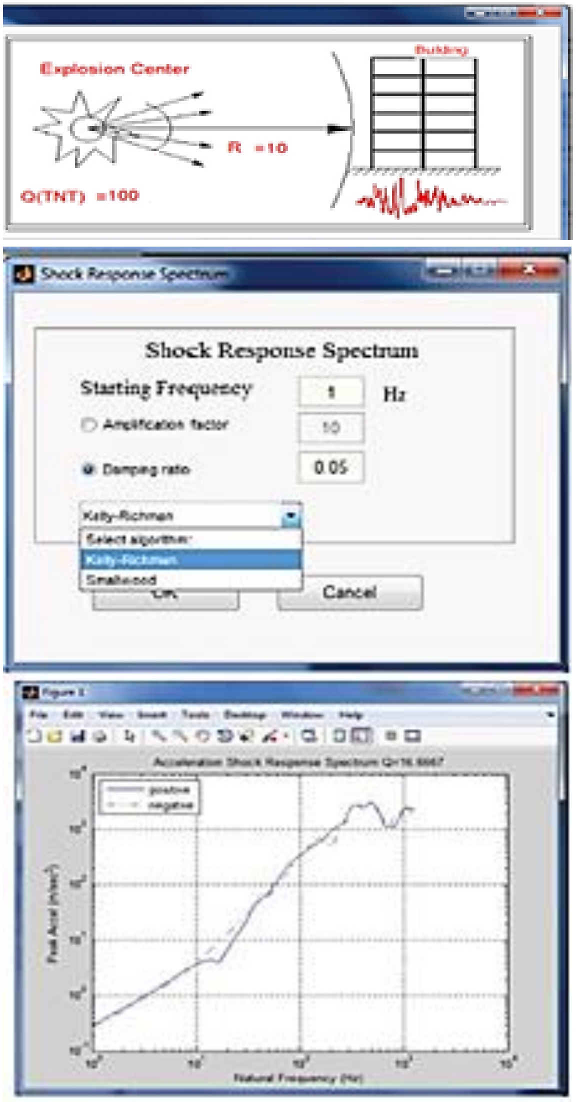


Fig. 10 Input information and results of shock response spectrum from the program

In accordance with Fig. 12, in order to apply an explosion load of 100 kg of TNT at a distance of 10 meters from the bath, the resulting acceleration was determined by BlastGM software at points A, B and C and in Abaqus software The instrument was applied.

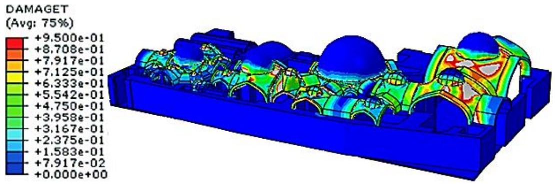


Fig. 11 Distribution of tensile damage of the bath in the final loading step

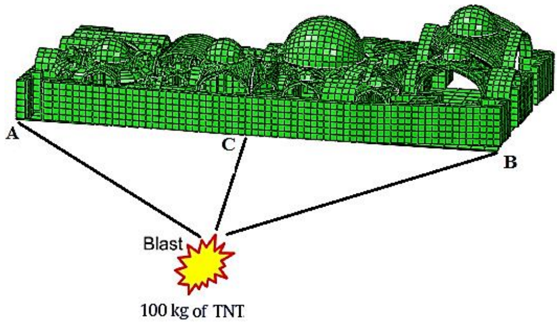


Fig. 12 Loading an explosion on Sheikh Baha'i's bath

The displacement-time curves of Sheikh Baha'i's bath under explosion for points A, B and C are shown in Fig. 13. According to the results of this figure, the maximum displacement occurred at point C in the center of the bath and Due to its proximity to the blast wave, the displacement of this point increased.

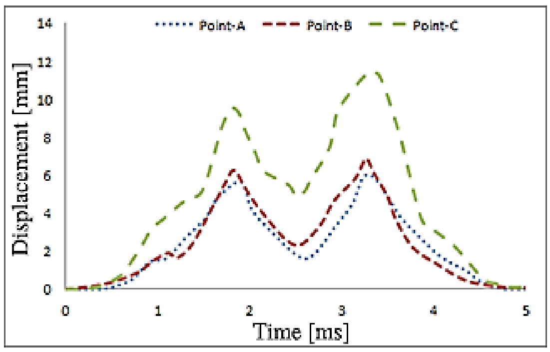


Fig. 13 Time history of the displacement under explosion

4- Conclusion

Due to the national and historical importance of Shiekh Bahaei's bath, in this study, the vulnerability of Shiekh Bahaei's bath under the Blast wave was investigated. Based on the results of this study, the location of the breakdown and the amount of displacement in different parts of the bath under the blast wave were determined and the most damage occurred at the location of the domes in the southern part of Sheikh Bahaei's bath which indicates the occurrence of failure in locations with higher altitude and greater weight. The results also showed that locations closer to the blast wave would accelerate more.

References

[1] Hinman, E. (2003). Primer for design of commercial buildings to mitigate terrorist attacks. *Risk management series FEMA. Federal Emergency Management Agency, Washington, DC*.

[2] Illampas, R., Charmpis, D. C., & Ioannou, I. (2014). Laboratory testing and finite element simulation of the structural response of an adobe masonry building under horizontal loading. *Engineering Structures*, *80*, 362-376.

[3] Valente, M., & Milani, G. (2016). Non-linear dynamic and static analyses on eight historical masonry towers in the North-East of Italy. *Engineering Structures*, *114*, 241-270.

[4] Tiberti, S., Acito, M., & Milani, G. (2016). Comprehensive FE numerical insight into Finale Emilia Castle behavior under 2012 Emilia Romagna seismic sequence: Damage causes and seismic vulnerability mitigation hypothesis. *Engineering Structures*, *117*, 397-421.

[5] Valente, M., Barbieri, G., & Biolzi, L. (2017). Seismic assessment of two masonry Baroque churches damaged by the 2012 Emilia earthquake. *Engineering Failure Analysis*, *79*, 773-802.

[6] Hejazi, M., Baranizadeh, S., & Daei, M. (2016). Optimum Shape, Failure Load and Buckling Load of Persian Historical Brick Masonry Domes. *Journal of Housing and Rural Environment*, *35*(155), 61-76.

[7] Akhvisi, A., Tavanaifar, H. (2015). Analysis of failure of unreinforced masonry structures based on strain hardening and softening in the framework of multi-plane model. *Numerical Methods in Engineering, 34* (1).

[8] Thai, D. K., & Kim, S. E. (2018). Numerical investigation of the damage of RC members subjected to blast loading. *Engineering Failure Analysis*, *92*, 350-367.

[9] Bozorgvar, M., & Shoushtari, A. (2012) Effects of explosion on earthquake-resistant concrete buildings. *Sixth National Congress of Civil Engineering*.

[10] Khandelwal, K., El-Tawil, S., & Sadek, F. (2009). Progressive collapse analysis of seismically designed steel braced frames. *Journal of Constructional Steel Research*, *65*(3), 699-708.

[11] Abroshan, F., and Moradloo, J. (2010). Dynamic analysis of geometric nonlinear behavior of composite shear walls under explosive loading. *Fifth National Congress of Civil Engineering*, Ferdowsi University of Mashhad, Mashhad, Iran.

[12] Shallan, O., Eraky, A., Sakr, T., & Emad, S. (2014). Response of building structures to blast effects. *International journal of engineering and innovative technology*, *4*(2), 167-175.