

Finite Element Analysis of Load Bearing Capacity of a Reinforced Concrete Frame Subjected to Cyclic Loading

Mahdi Bamdad^a, Abdolreza Sarvghad Moghadam^b, Mohamad Javad Mehrani^{c*}

^a Department of Engineering, Islamic Azad University, South Tehran Branch, Tehran, Iran.

^b Associate Professor, International Institute of Earthquake Engineering and Seismology, Tehran, Iran.

^c Department of Civil and Environmental Engineering, Shahid Beheshti university, Tehran, Iran.

Received 15 April 2016; Accepted 20 May 2016

Abstract

Many methods have been developed in order to study the impact behavior of solids and structures. Two common methods are finite element and experimental method. The nonlinear finite element method is one the most effective methods of predicting the behavior of RC beams from zero-load to failure and its fracture, yield and ultimate strengths. The advantage of this method is its ability to make this prediction for all sections of the assessed RC beam and all stages of loading. This paper compares the experimental results obtained for a RC frame with the numerical results calculated by ABAQUS software, and plots both sets of results as hysteresis–displacement diagrams. This comparison shows that the numerical FEM implemented via ABAQUS software produce valid and reliable results for load bearing capacity of RC frames subjected to cyclic loads, and therefore has significant cost and time efficiency advantages over the alternative approach

Keywords: ABAQUS; Reinforced Concrete Frame; Displacement Force Diagrams; Pushover Analysis.

1. Introduction

Earthquakes around the world have shown the importance of the rehabilitation of existing buildings; especially those were built before the modern codes of seismic design were issued [1]. Many traditional methods have been used for strengthening the RC structures such as adding of RC infill walls, precast panels, steel bracing, and concrete jacketing of the frame member [2]. Simulation of impact behaviour of solids and structures still poses significant difficulties on computational methods and constitutive models [3]. Finite element method is the numerical approach which is used to solve approximately partial differential equations[4]. The reinforced concrete (RC) moment-resisting frames with masonry infill walls are widely used in buildings. It has been well recognized that the arrangement and constructional detail of infill walls have significant effects on the seismic performance of RC frames [5]. The “behavior factor” is widely recognized as the most important parameter of seismic design. The nonlinear finite element method is one the most popular and effective methods of assessing the exact behavior of RC beams from zero load until failure, and obtaining its fracture, yield and ultimate strengths. The advantage of this method is its ability to predict the behavior of all sections of the assessed RC beam at all stages of loading [6]. ABAQUS is finite element software with extensive use in engineering applications, mostly because it lacks the flaws of other software developed for this purpose. This software consists of three main components: i) ABAQUS/Standard for solving all linear and nonlinear static and dynamic problems, ii) ABAQUS / Explicit for modeling the transient dynamic problems such as collisions, impacts as well as quasi-static problems, and iii) ABAQUS/CAE, which is a GUI designed to facilitate the procedure of defining the model, the boundary conditions, and the loading process. In a study by Bolea (2016), author used the laboratory of University of Bucharest to examine the seismic response of RC frames with masonry infill

* Corresponding author: mj.mehrani@gmail.com

panels [7]. Jiang et al. (2015) studied the seismic behavior of RC frames with masonry infill panels under cyclic loads. They analyzed the influence of constructional details of infill walls on the seismic behavior of RC frames. It is found that with the addition of masonry infill wall rigidly connected to the frame, the lateral strength, the stiffness and the energy-dissipation capacity of the bare RC frame increase significantly while the displacement ductility ratio decreases significantly [5]. The study conducted by Shafei et al. (2013) assessed the effects of flexible joints on the lateral response of reinforced RC frames. Seismic effects were modelled in the OpenSees software framework and a modified joint element for analysis of multi-storey frames was used [8]. In the study conducted by Mondal et al. (2013), authors assessed the behavior factor of 2, 4, 8 and 12 story RC structures. Their research focuses on estimating the actual values of 'response reduction/modification factor' (R) for realistic RC moment frame buildings designed and detailed following the Indian standards for seismic and RC designs and for ductile detailing, and comparing these values with the value suggested in the design code [9]. Piera. (2005), also studied the Performance evaluation of masonry-infilled RC frames under cyclic loading based on damage mechanics. Damage model is proposed in his paper for the characterization of masonry walls submitted to lateral cyclic loads. The model includes the simulation of phenomena such as stiffness and strength degradation and pinching behavior. The macromodel has been incorporated in a nonlinear structural analysis program for analysis of masonry-infilled RC frames [10].

Experimental modeling, detailed analysis, and numerical solution methods are some of the most effective methods of solving physical problems. The major weakness of experimental methods is the costly and time consuming nature of their procedures, but numerical methods have proven to be great alternatives in this respect. Conventional numerical methods have also shown great potential in solving the problems with complicated boundary conditions, something that detailed analyses often fail to accomplish. In this study, we will compare the experimental and numerical methods to assess the validity of numerical finite element method implemented via ABAQUS software.

2. Research Methodology

This paper uses the numerical finite element method implemented via ABAQUS (Version 6.12) to study a RC frame model. To validate the finite element model, we use ABAQUS to simulate a recently performed laboratory experiment [11], on the RC frame and compare the simulation results with the actual values reported by the reference. The specification showed in figure 1 is used to create the model as it is described in the "Parsa and Sarvghad Moghadam [11]". It should be noted that all geometric parameters of this finite element analysis are defined in accordance with the experiment conducted in this reference [11]. Some parts of the geometric model are partitioned to provide better meshing and more regular elements. The resulting geometric model is then imported into the "Mesh module" to undergo the meshing procedure, producing the FE model. The following figures show the geometry of the studied RC frame and its finite element model generated in the ABAQUS software.

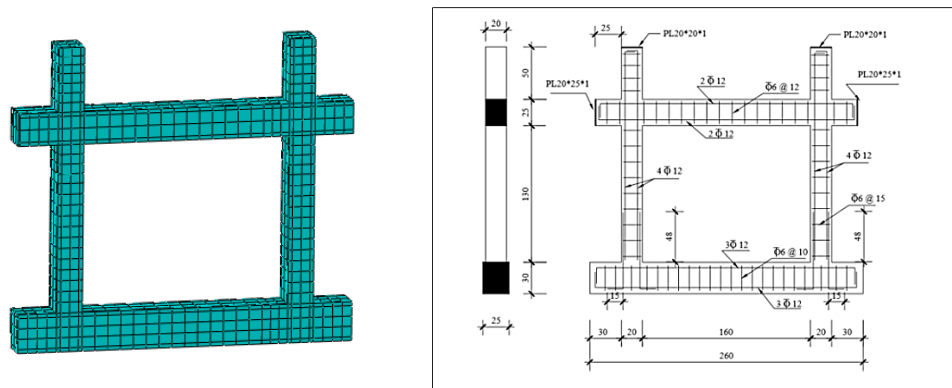


Figure 1. Geometric characteristics of the chosen RC frame [11] (right), and its finite element model created in ABAQUS software (left)

Table 1. Geometric characteristics of the studied RC frame as described by [11]

Section	Dimensions	Longitudinal bars	Transverse bars
upper beam	20×25	12Φ2	12@6Φ
Lower beam	25×30	12Φ2	12@6Φ
Column	20×20	12Φ4	15@6Φ

The assessed moment-resisting RC frame is subjected to an increasing displacement controlled cyclic load applied along its upper beam. The displacement load is applied horizontally in the form of cyclic hysteresis on the top section of the left side of the beam, which is analogous to the load exerted by a hydraulic jack.

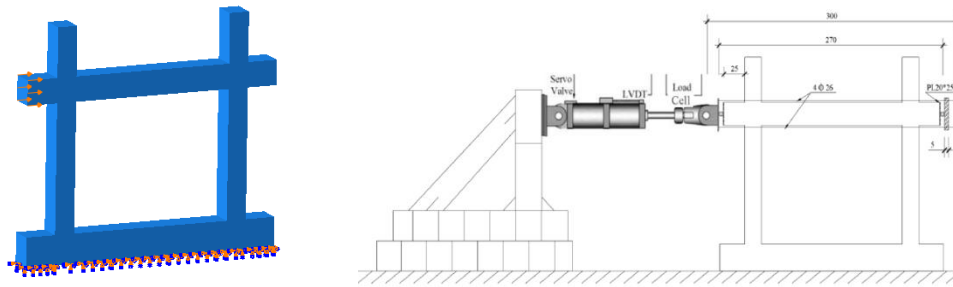


Figure 2. Load and boundary conditions used in the laboratory experiment [11] (right), and the load and boundary conditions used in ABAQUS simulation (left)

3. Results

Diagrams resulting from experiment and ABAQUS simulation show similar behaviors, which indicate a good agreement between the results of FEM analysis and the actual values reported by the Parsa and Sarvghad Moghadam [11]. According to the reference, the maximum load the RC frame can sustain is 4 tonnes, while the results given by ABAQUS predict this parameter as 4.42 tonnes. This shows a 10.60% difference in maximum lateral force sustainable by the RC frame, which point to a relatively good precision. This result validates the numerical FEM method implemented via ABAQUS software. The following figures show the evolution of the load (Figure 3) and the behaviour of moment-resisting RC frame (Figure 4).

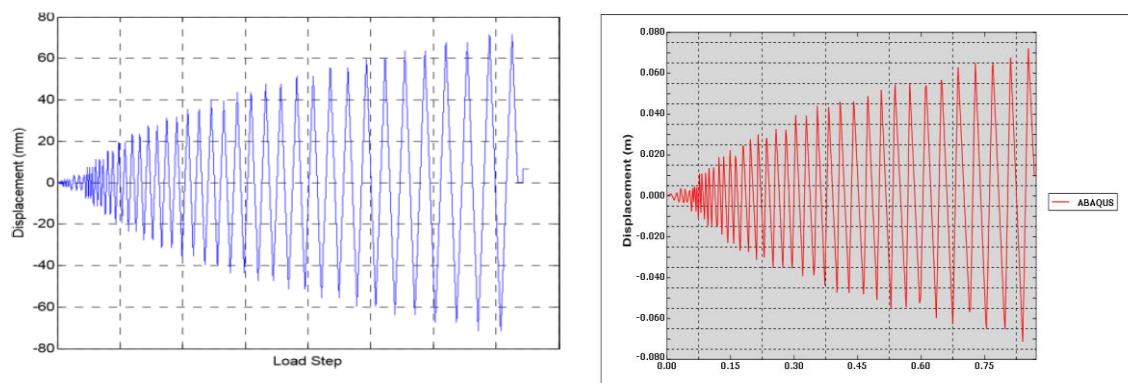


Figure 3. Loading history given by Parsa and Sarvghad Moghadam [11] (left) and this study (right)

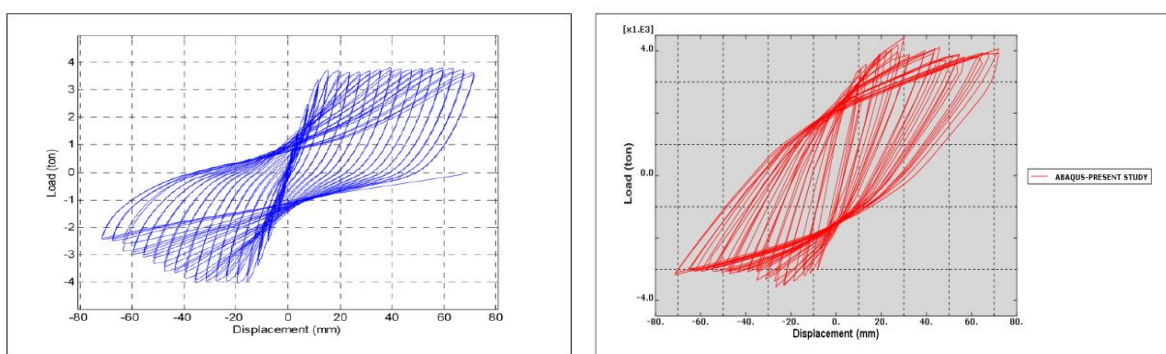


Figure 4. Hysteresis diagram of the RC frame given by Parsa and Sarvghad Moghadam [11] (left) and this study(right)

Hysteresis curves obtained by numerical study conducted through FEM show that the maximum sustainable force of this RC frame is about 4.42 tones. These results also show clearly reduced stiffness and also declining strength after the displacement of 4 cm. Deterioration of strength and stiffness, which is caused by the repeating loops of cyclic load, accelerates with the further damage of RC frame. Once lateral displacement is applied, RC frame starts to exhibit flexural cracks in the upper sections of the columns. As the magnitude of this displacement and the number of cycles increase, these cracks undergo further development, ultimately leading to formation of plastic hinges at topmost sections of both columns. It is also observed that the damage caused by the cracks formed in columns' lowermost sections increases with the increasing number of cycles and can ultimately lead to full separation. Figures 5 to 7 show

the contours of damage caused by tensional crack and their development in different stages of loading simulated by ABAQUS software.

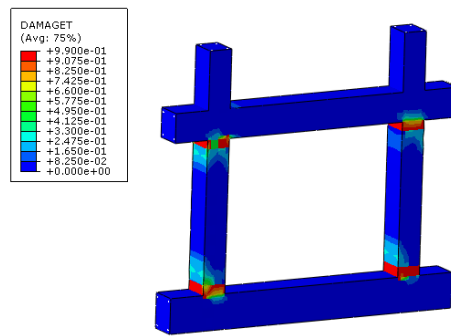


Figure 5. Damage caused by cracking in the initial loading cycles

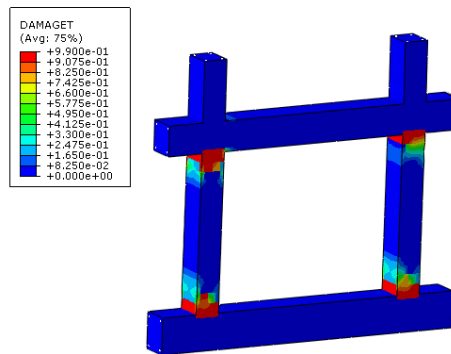


Figure 6. Damage caused by cracking in the intermediate loading cycles

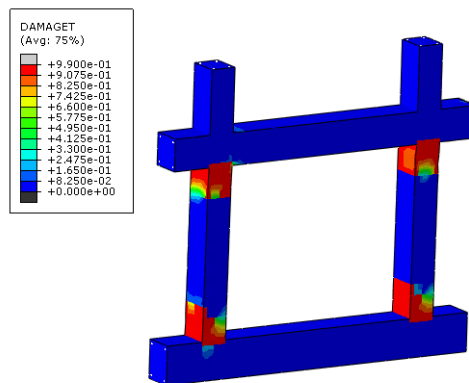


Figure 7. Damage caused by the formation of plastic hinge at the columns' topmost section and separation of their lowermost section from the foundation at the late loading cycles

4. Conclusion

According to the results obtained from experiment with those obtained by numerical FEM (Figures 3 and 4), the maximum sustainable load of RC frame obtained by the reference is 4 Tones, and the numerical FEM implemented via ABAQUS software predicts this parameter as 4.42 Tones. This shows a 10.60% difference in maximum lateral force sustainable by the RC frame, which indicates a good precision. Figures 5 to 7, which are the outputs of ABAQUS software, show a clearly reduced stiffness coupled with declining strength after the displacement of 4 cm. Declining strength and stiffness caused by the repeating loops of cyclic load accelerates with the further damage of RC frame. Applying the lateral displacement causes the RC frame to develop shallow flexural cracks in the columns' upper sections, formation of plastic hinges, and separation of column feet from the foundation. Overall, it can be said that the numerical FEM implemented via ABAQUS software produce valid and reliable results for load bearing capacity of RC frames subjected to cyclic loads.

5. References

- [1] Batikha, M. and F. Alkam, *The effect of mechanical properties of masonry on the behavior of FRP-strengthened masonry infilled RC frame under cyclic load*. Composite Structures, 2015. 134: p. 513-522.
- [2] Rai DC, *Seismic rehabilitation of structures. India: Department of earthquake engineering*. 2009, University of Roorkee.
- [3] Kumar, V. and K. Ramamurthy, *Meshfree and finite element modelling of impact: A comparative study*. International Journal of Impact Engineering, 2016. 90: p. 146-153.
- [4] Balomenos, Genikomsou, Aikaterini S, Polak, Maria Anna Pandey, Mahesh D, *Efficient method for probabilistic finite element analysis with application to reinforced concrete slabs*. Engineering Structures, 2015. 103: p. 85-101.
- [5] Jiang, H., X. Liu, and J. Mao, *Full-scale experimental study on masonry infilled RC moment-resisting frames under cyclic loads*. Engineering Structures, 2015. 91: p. 70-84.
- [6] Naderpour KH, *nonlinear behavior of reinforced concrete beams with high strength*. Ferdowsi University of Mashhad Civil Engineering Journal, 2010. 21.
- [7] Bolea, Ovidiu, *The Seismic Behaviour of Reinforced Concrete Frame Structures with Infill Masonry in the Bucharest Area*. Energy Procedia, 2016. 85: p. 60-76.
- [8] Shafaei, J Zareian, M.S Hosseini, A Marefat Sadegh M, and M.S. Marefat, *Effects of joint flexibility on lateral response of reinforced concrete frames*. Engineering Structures, 2014. 81: p. 412-431.
- [9] Mondal, Apurba Ghosh, Siddhartha, and Reddy, *Performance-based evaluation of the response reduction factor for ductile RC frames*. Engineering Structures, 2013. 56: p. 1808-1819.
- [10] Ricardo Perera., *Performance evaluation of masonry-infilled RC frames under cyclic loading based on damage mechanics*. Engineering Structures, 2015. 27: p. 1278-1288.
- [11] Parsa F, Sarvghad Moghadam A., *Experimental studies in the frame Infill Building materials Concrete*. Journal of Tehran University, College of Engineering, 2009. 42(6).