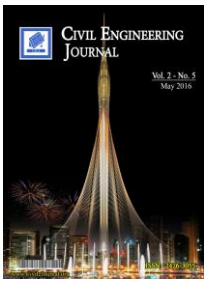
Available online at www.CivileJournal.org

Civil Engineering Journal

Vol. 2, No. 5, May, 2016



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