Engineering and Technology Journal
Journal homepage: engtechjournal.org

Nonlinear Finite Element Analysis of the Seismic Retrofitting of Existing Buildings

Shirin A. Abdulla [a*], Mohammed J. Hamood [b], Mohammed A. Elaiwi [c]

[a] Civil Engineering Dept., University of Technology, Baghdad, Iraq. 42155@student.uotechnology.edu.iq

[b] Civil Engineering Dept., University of Technology, Baghdad, Iraq. 40040@uotechnology.edu.iq

[c] Civil Engineering Dept., University of Technology, Baghdad, Iraq. 40080@uotechnology.edu.iq

*Corresponding author.

Submitted: 15/09/2020 Accepted: 08/11/2020 Published: 25/06/2021

K E Y W O R D S

Connection Details,

Finite Element Analysis, Numerical Simulations, RC Infill Walls, Retrofitting of RC buildings, Abaqus (2019) software.

A B S T R A C T

Finite Element Analysis (FEA) is a helpful tool for finding solutions to civil engineering problems. In this paper, the numerical analysis was performed by simulating an experimental test by using FEA tools, Abaqus /CAE2019 was used to develop and retrofit solutions to sustain existing structures for seismic hazard. The 3D building was modeled, and nonlinear analysis was adopted. This experimental building was tested at the Joint Research Centre (JRC) of the ELSA facility that is in Ispra-Italy. This model was a building of a full-scale four-story tested using the pseudo-dynamic (PsD) techniques.

The retrofitting model was done by adding new RC walls with different connection details to the existing building. This building corresponding in gravity design only. The goal of the experimental test was to study the effectiveness of adding RC infill walls as retrofitting method, including designing it and the contribution of dowels that connect the new infill wall to the existing RC building. In other words, it is a strengthen method carry out by conversion of selected bays into new infilled RC walls.

The results of analytical modeling of the RC structure in the Abaqus software show that the percentage of differences of X- Direction in top story displacement between Abaqus software and Experimental tested at ELSA results are 2.47% in positive and 3.12% for negative X direction, which refer to a very good similarity and accurate building simulation.

How to cite this article S. A. Abdallah, M. J. Hamood, and M. A. Elaiwi, “Nonlinear Finite Element Analysis of the Seismic Retrofitting of Existing Buildings,” Engineering and Technology Journal, Vol. 39, No. 06, pp. 915-927, 2021.
DOI: https://doi.org/10.30684/etj.v39i6.1834

INTRODUCTION

Finite Element Analysis is a helpful tool for finding solutions to civil engineering problems. It is fundamentally a process where a continuum with unlimited degrees of freedom in which corresponds to be a congregation of elements (or sub-regions) each with an unknown specified finite numbers. This substantial simplicity in both mathematical form and physical interpretation has been certainly behind its popularities as much was the digital computer, which allows in the current time a realistic solution of more complicated engineering cases [1]. Accordingly, as the multi-story RC buildings, dynamic analysis equations gating more complex and with high difficulty in solving, therefore, the numerical solution is proposed to be used. To obtain an accurate result from the modelling of the structure, several initial assumptions needed to be made and validated.

Abaqus /CAE 2019 software is used in this paper [2], for validation of practicality, by simulating full-scale building has been constructed and tested at the Joint Research Centre (JRC) of the ELSA facility that is in Ispra-Italy. The purpose of these experimental works was to study the effectiveness of the proposed retrofit methods by converting selected bays into infill RC walls. Details can be found in [3], [4] and [5].

1. OBJECTIVE

This paper provides a depiction of the processes of the finite element formulation as the following:

1. Validate the use of numerical simulation of modelling a building that has been experimentally tested in the Elsa Laboratory in Ispra (Italy) to study the effect of adding RC infill walls in the middle of the frame, which is one of the seismic retrofitting methods under dynamic load conditions.
2. Study the non-linear behavior of a four-story building, a building that designed to resist gravity loads only, accordingly, has some features that are different from those buildings constructed by codes that contain the seismic design. Many buildings in Iraq are designed to sustain gravity load only. These properties cause deficiencies in building response to seismic loadings.

2. PREVIOUS STUDIES

An experimental study was conducted on the seismic modernization and strengthening of existing buildings. This study was carried out using large-scale PsD techniques, four-story RC frame with RC wall filling by converting targeted bays into new RC infill walls using RC infilling at the ELSA facility in Ispra-Italy. It was designed based on the BS8110 (1983) [6] and Euro code2 [7]. The building was (12.0 m high), three-bay (8.50 m long) with parallel frames connected by 0,150 m slabs with RC wall infilled central bay (2.50 m) were tested with pseudo-dynamic method. The existing frames detailed and designed for only gravity load resistance and were similar to frames constructed in the 1970s in Cyprus. Different reinforcement percentages and connection details for the two infilled frames were selected with the purpose of studying their influences in measuring building response. The results of the cyclic and pseudo-dynamic tests show that, the building managed to tolerate a seismic load of 0. 25g with no significant damage, also it was proven that adding infill wall into selected bay can be used to increase existing strength, ductility and reduce structures deficiencies. The structure recorded local and global behavior provides data for numerical model development, accordingly, design guidelines was proposed for such a retrofitting method [3] and [4].

Another study performed a numerical simulation of the experimental seismic results that used for strengthening of existing structures. An experimental campaign that studied the effectiveness of seismic retrofitting of existing structures was simulated and analyzed in DIANA finite element analysis (FEA) software. The retrofitting method that was studied is the conversion of selected bays into new infilled RC walls for the retrofitting of a multi-story multi-bay RC frame building. The building was tested at the Joint Research Centre in the ELSA facility located in Ispra-Italy. It was a full-scale four-storey model tested with the pseudo-dynamic (PsD) method. The existence frames designed and detailed for only gravity load resistance by using different connection details between the bounding frame and walls. The purpose of the experimental tests was to investigate the efficiency of the suggested retrofitting technique, designing of new infill walls, and the dowels contribution that connect the existing RC frame to the new infill wall. Nonlinear transient analyses were performed in addition to 2D FEA frames modeling with the aim of simulating the experimental results. It was found out that, the 2D finite element model numerical results could simulate the behavior of the test specimen with high accuracy. Accordingly, the complement of the experimental results can be performed and the interaction between the RC infills frame and surrounding both in the global and local level can be investigated. These results will contribute to the studying of an application of the general model in addition to designing RC infills in existing RC frames. [8]

There is a study on conducting a numerical simulation to investigate the efficiency of the RC frame building seismic modernization by shifting a selection of bays to new walls by RC in sap software program. This studied includes tested four-story model of the Pseudo dynamic method. The frames designed for only gravity load resistance. The main objective of this study is to clarify the model's capabilities to simulate the experimental non-linear behavior of the Laboratory Coordinator, Resident buildings, which reinforced using RC filling walls and to observe on their efficiency. The experimental results are associated with the damage of widespread noted during the simulation with the purpose of extracting a hybrid fragility curve framework. The comparison of the building fragility curves before and after the retrofitting offers vital data concerning the reduced probability of high damage and collapse. The enhancement in the building dynamic characteristics after the retrofitting was also involved in the framework. It observed from the fragility curves that, a considerable ductility amount accomplished prior to the building collapse due to the RC Infill walls insertion and this considered as a success of the retrofitting method. [4] and [5].

4. METHODOLOGY AND MATERIAL

I. Description of the geometrical model

1) The model consisted of four stories with two external frames named (north and south) frames contain three bays detailed in SERFIN Project. [10]
2) These frames were spaced at 6.0 m and are connected to a 0. 15m RC solid slab and four beams (0.25m by 0.50m) perpendicular to the plane of the two frames three-bay.
3) The building total length was 8. 5m (Central Bay 2.5m and two 3.0m outer bays) with 3. 0m floor height and 12m total building height as shown in Figures 1 and 2.
4) The columns were 0. 25m from 0.4m.
5) The building was constructed on an 11.0 by 8.0 m foundation slab with a 0. 40m thickness, with 0. 4m high and 0. 6m wide beam.
6) The RC infill walls were placed in the central bays of the building with the same thickness as 0. 25m as columns and beams surrounding them.
7) CFRP Location at the bottom of a ground shear wall column, connection to the north frame.

II. Assumption in the experimental work and in the numerical study

As follows, some assumptions, which were considered in this investigation in the original design that was adopted:

1) The concrete design was based on the BS8110 (1983) revision code.
2) The adopted building was designed to resist gravity loads only and consequently has properties that be different from those in typical buildings constructed to sustain seismic loads. These properties source deficiencies in building response under seismic loadings.
3) Friction between soil and RC is not considered (fixed base was considered).
4) Earthquake acceleration in X-direction of has been adopted according to the location of actuators in the experimental test.
5) The influence of pore water pressure was considered.
6) The effect of the CFRP has been considered to reduce the seismically hazard.
7) The effect of wind load was neglected.

III. Materials used in the experimental test

The materials used in the model were based on their availability in the Italian laboratory. It was decided to adopt C20/25 concrete for both the walls and the frame with a unit weight of 25kN/m³ and E = 30GPa modulus of elasticity. The yield strength of reinforced was 420 MPa While the yield strength of the reinforcement of infill wall consisted starter web bar and impeded rebar was 450 MPa, this material used in construction practice in Cyprus in the 1980s. Figure 3 illustrated materials properties. This building was numerically analysis using three-dimensional finite element models by adopting non-linear material behavior. The response of RC structures can be achieved by accurately modeling the stress/strain behavior of uniaxial materials. Thus, table (1) and (2) shows the results of the ELSA laboratory test. [11].

5. EXPERIMENTAL MODEL DESIGN

The model is an expression of the buildings that prevailed in the 1980s in Cyprus. At that time, the structures were designed for gravity loads only because there were no codes includes seismic loads. Therefore, there was no standard for seismic design. Accordingly, it was decided to use the BS8110 code, which were very close to the CP110 code with minor differences.

The self-weight was calculated using the unit weight of concrete specified above. Each floor was supposed to be loaded with 3 k/m2 of dead load (include the load of masonry infill walls) and 30% of 1.5 kN/m2 live loads. Thus, (1.0 x 3.0 kN/m2 + 0.3 x 1.5 kN/m2) x 6.25 m x 8.90 m = 192 kN that applied on each floor.135.4kN was applied with 15 barrels of water and the rest was the self-weight of the engine attachment packages. (Figure 4 showed barrels of water)

To simulate the actual condition where the existing building is retrofitted, the building was loaded with dead load before the casting of the infill walls while, live load added after casting.

These loads were combined using partial factors of safety of 1.40 for self-weight and imposed dead-load, and 1.60 for the live load. All details of building reinforcement are shown in table 3 and figures (5 to 9).

To use the case study for more various parameters, the sample (test building consist of two frames north frame and south frame were reinforced in a different amount and location for reinforcements, the northern wall being the strongest between the two. More specifically, a detailed and irregular system of dowels and starter bars was used to join the walls with the frame are shown in figure (10). It is important to mention that the tested model was designed using two different connection details between the new walls and the surrounding frame to evaluate the contribution of dowels that connect the new infill wall to the existing RC frame, table ٥, 6 and figure 10 illustrations the different types of the dowels.

6. REPRESENTATION OF SEISMIC LOADS IN EXPERIMENTAL WORK

The recorded (Psd) using one-directional loadings based on a ground motion recorded at Herceg-Novi station throughout the Montenegro earthquake that happen in 1979 as shown in Figures (11) and (12). A record was applied to the structure during three runs of the linearly increasing intensity of the peak ground acceleration (PGA), such as:

1-0. 1g PsD test
2- 0.25g PsD test.
3- Cyclic test

The 0. 1g test was initially performed to include minimal damage to the building. Then, at 0. 25g test was conducted to study the building performance at its final capacity. For the final periodic test, a displacements history was applied on the fourth floor, with a triple load distribution maintained over the height of the north frame with zero rotation on each of the four floors (one-way test).

7. VALIDATION OF THE PROBLEM

I. Elements used in ABAQUS software

The validation of the developed simulation model is significant. Abaqus/Cae 6.19 software is used in this investigation regarding the effect of dynamic vibration. The numerical results obtained from this computer software were compared to experimental results of a full-scale RC structure with four story, three multiple one bays. The test was conducted at the European Laboratory Joint Research Center specifically in ELSA that located in Ispra, Italy.

To model the concrete elements solid (brick C3D8R/8-node) are used, however, linear (truss T3D2/2-node) are used to model the rebar element while the (CFRP) sheets used (S4R - 4-node doubly curved. These elements are multipurpose and can be applied in models for direct linear or complex nonlinear analyses, including plasticity of properties, large deformations, and contact. The typical solid elements in Abaqus are shown in Figure 13 (a), (b) & (c). The numerical reinforced concrete model is shown in Figure 14.

II. MATERIALS MODELING IN ABAQUS

Material modelling is an important part of Finite Element Analysis. The Abaqus software, with its emphasis on nonlinear FEA and large deformation analysis, has provided advanced material models since its inception. For example, the Concrete Damaged plasticity model (CDPM) offers the tool to accurately simulate the (reinforced or plain) concrete element's behavior under dynamic load. The CDPM allows for the stiffness recovering during reversals of the load.

III. The concrete damage plasticity model parameters

Performance can be described by other parameters measured for uniaxial stress. Table 5. Illustrated the parameters of the model that describe its performance to sustain compound stress.

IV. Mesh system and type element

First order Three-dimensional reduced integration continuum elements (C3D8R/8-node linear brick) are used to model the concrete members while the rebar is modeled by using (T3D2/2-node linear 3-D truss) element.

In the FEA modeling, the concrete block of the footing, slab, column, and beam are idealized for homogeneous material and modeled with solid (brick) eight-node elements, which recognized in Abaqus as C3D8R elements. The C3D8R elements selected because with reduced integration follow the constitutive law integration accurately and very suitable for nonlinear dynamic implicit analyses as well as allow for finite strain and rotation in large-displacement analysis. The modeling of longitudinal and lateral steel bars were as embedded elements through the concrete block with (3-D truss T3D2/2-node linear) element. Moderately fine mesh was chosen for constructing the model mesh, the provide a similar response to the results of the experimental work. The recommended rectangular mesh is used to acquire flawless results from the C3D8 element. It should mention that the mesh was prepared and rectangular or square elements were shaped in a way that, the elements length and width in the plates must be consistent with the nodes and elements in the models concrete portions. The finite element model of the reinforced concrete building with a typical mesh discretization of the concrete and steel rebar is shown in Figure 15.

8. VALIDATION RESULTS

The building pseudo-dynamically experimentally tested with a ground motion-based acceleration measured at Herceg-Novi station during Montenegro earthquake in 1979. One directional record was applied to the building in three linearly increasing intensity runs of the maximum peak ground acceleration, such as 0.10 g, 0.250 g and final cycling test. For numerical study, the identical same motion input was applied to verify the current analytical model adequacy. Figures (16 to 18) illustrated validation job, displacement in X direction and Maximum principal stresses. Table 6 and figures (19 to 22) Show floors displacement comparison between Abaqus program results and experimental results in ELSA laboratory, with the purpose of using the same model for further case studies with assurance.

Typical validation is of great importance in the advance of a simulation model. Because of the difficulty of conducting these tests in a practical way, moreover, no algorithm exists to select what procedures or techniques to be used. Each simulation project offers a unique and new challenge to developing the model.

The modeling of the reinforce concrete structure in the Abaqus software in table 6. Shows that, the percentage of differences in top story displacement in X- Direction between ELSA test and Abaqus software results are 2.47% in X for positive and 3.12% for negative top story displacement which refer to a very good similarity and accurate building simulation.

9. FIGURES

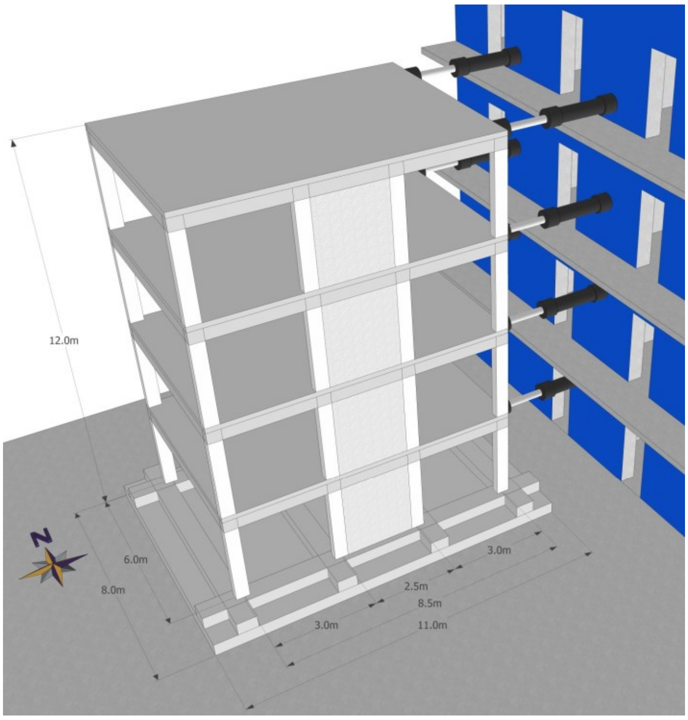


Figure 1: Full-scale of the building

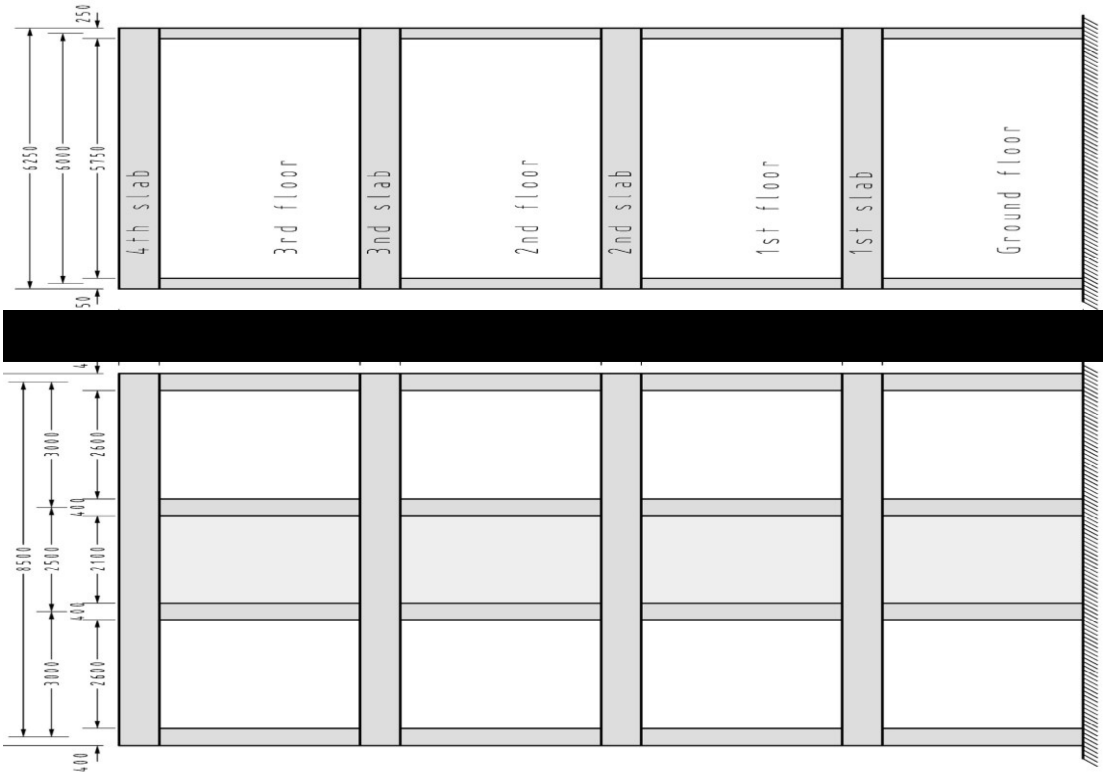


Figure 2: SERFIN specimen Side view with infill wall test model.

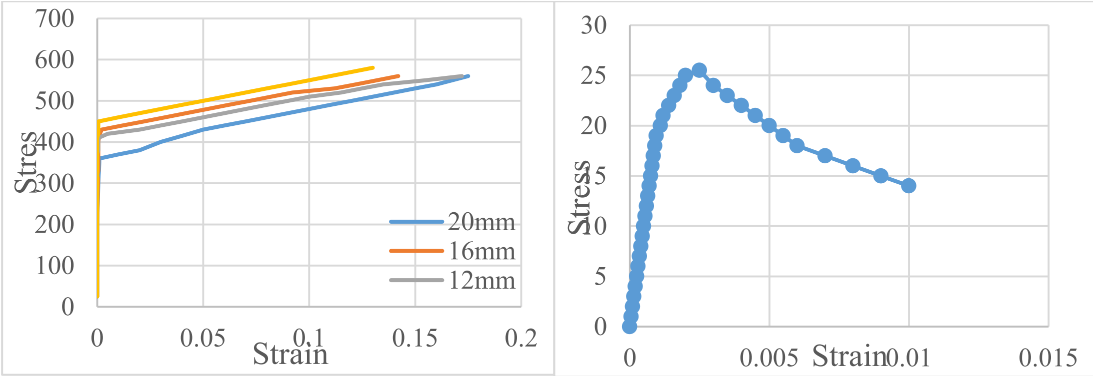


Figure 3: Stress- strain relationship for steel and concrete.

Figure 4: Barrels of water use in the building

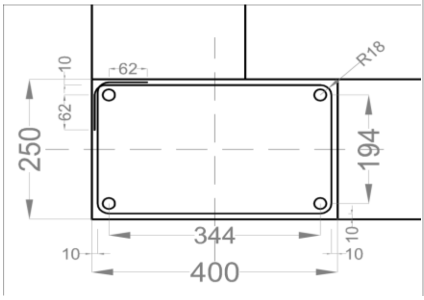


Figure 5: Column cross section

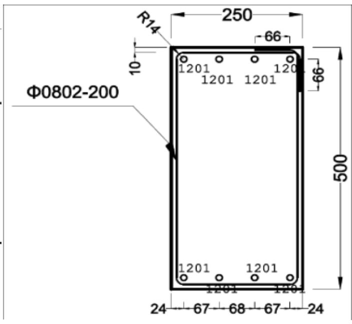


Figure 6. Longitudinal beam

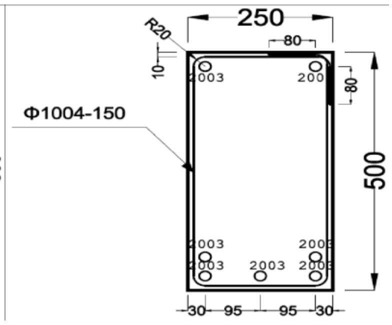


Figure 7. Transverse beam

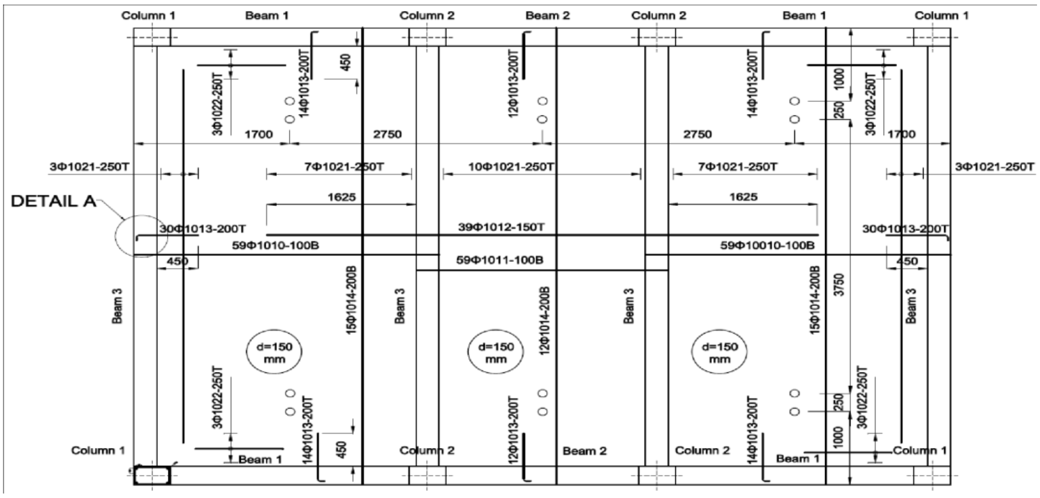


Figure 8. Reinforcement of slab, T refers to top and B refers to the bottom

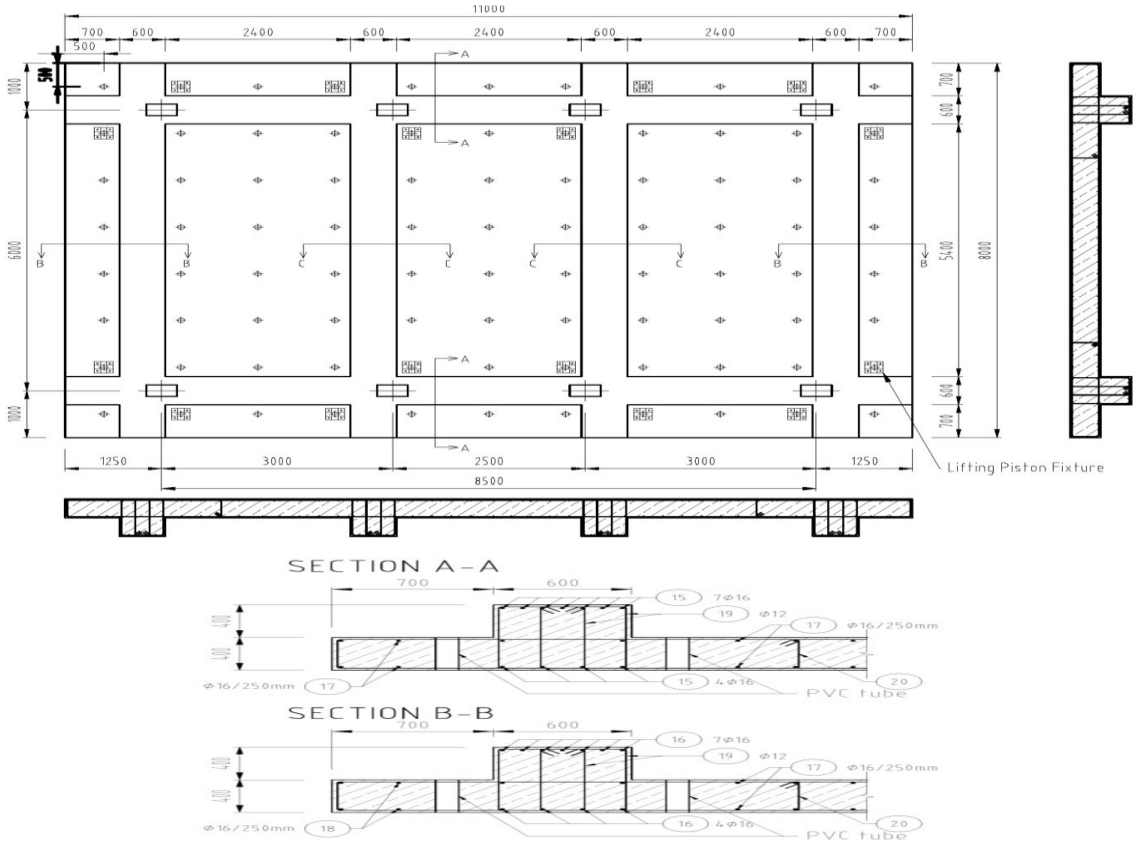


Figure 9. Foundation slab and sections.

(a) Short dowels **(b) starter and dowel bar.** **(c) Starter, dowels and web bars**

Figure 10. (a) Short dowels, (b) starter and dowels bar, (c) Starter, dowels and web bars

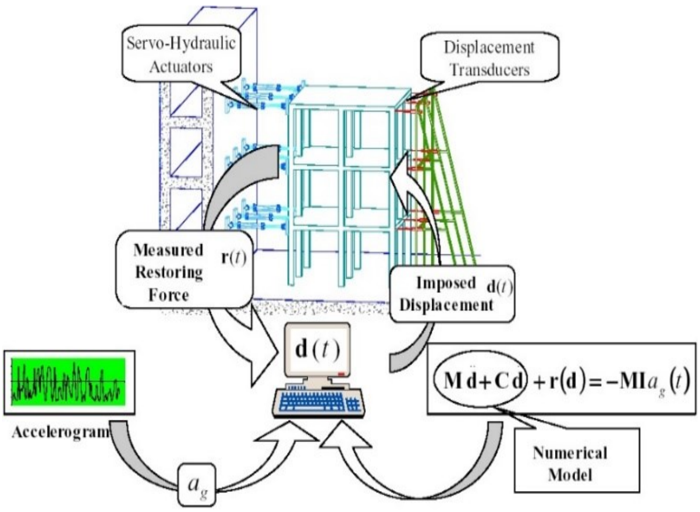


Figure 11. Pseudo-dynamic test method.

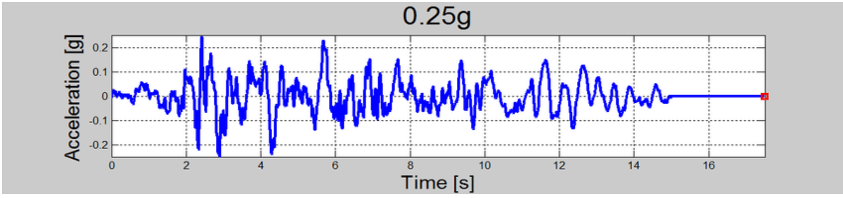
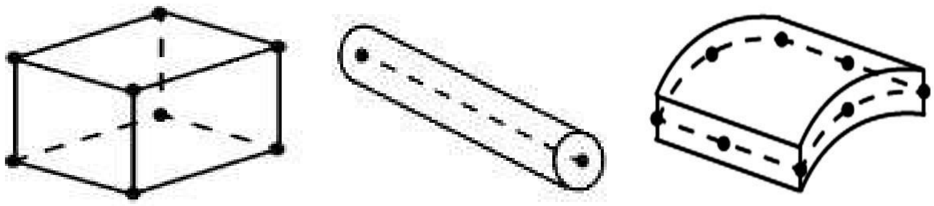


Figure 12. 0.25g of Herzeg Novi (Montenegro 1979).



(a) (b) (c)

a) Discretized of concrete using Brick Elements
b) Discretized of steel Reinforcement using truss elements
c) Discretized of CFRP sheets using Shell elements

Figure 13. (a), (b) and (c) Types of Elements.

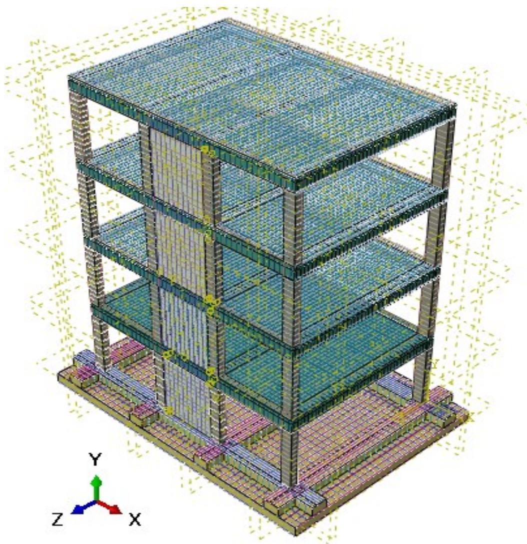


Figure 14. Construction Geometry Model That Used In Abaqus 6.19 Program

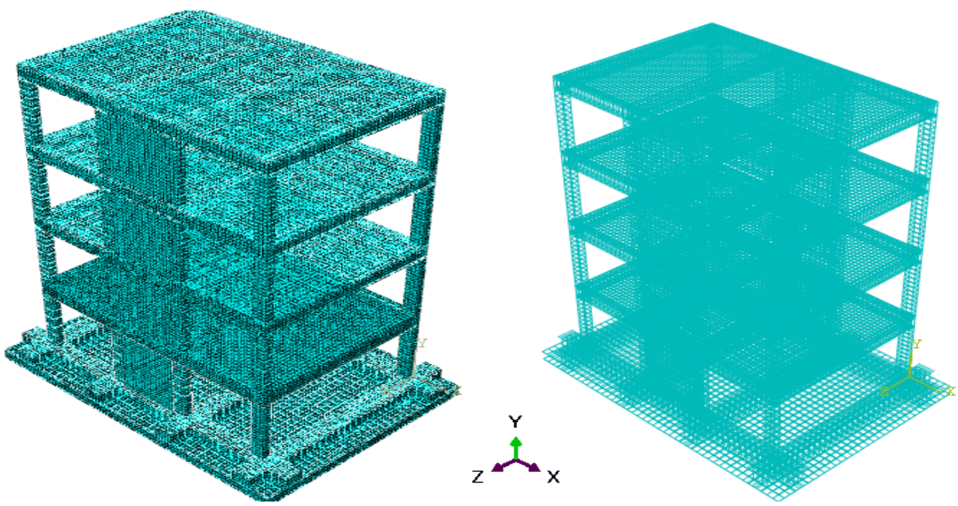


Figure 15. Finite Element Model For Concrete And Reinforcement In Abaqus6.19 Program

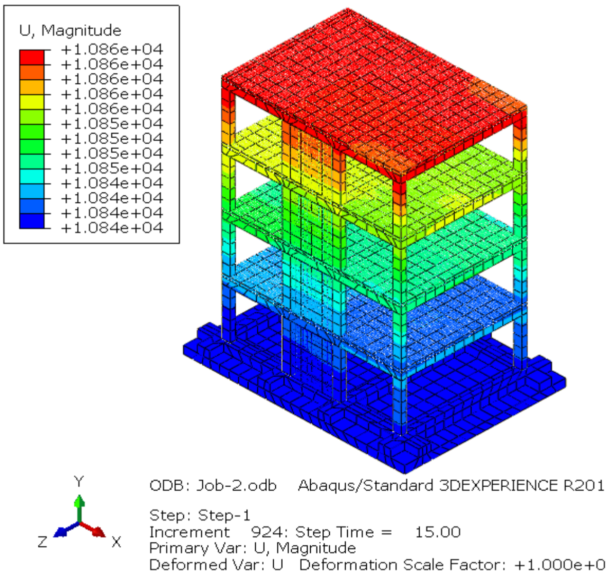


Figure 16. Displacements for all storeys for the model in the (X- direction)

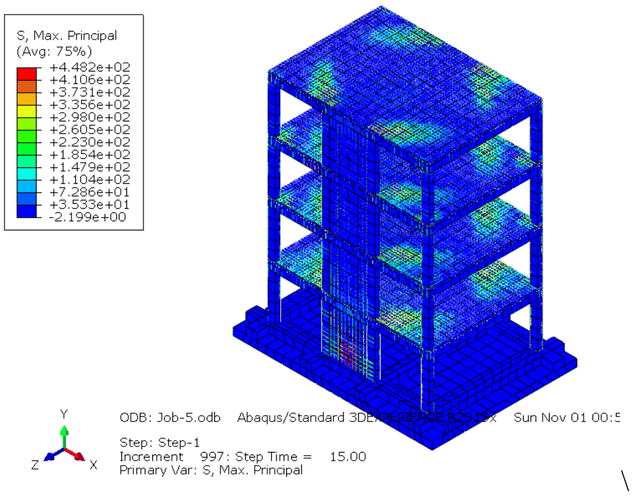


Figure 17. The max principal stresses in the Reinforced Concrete building for model

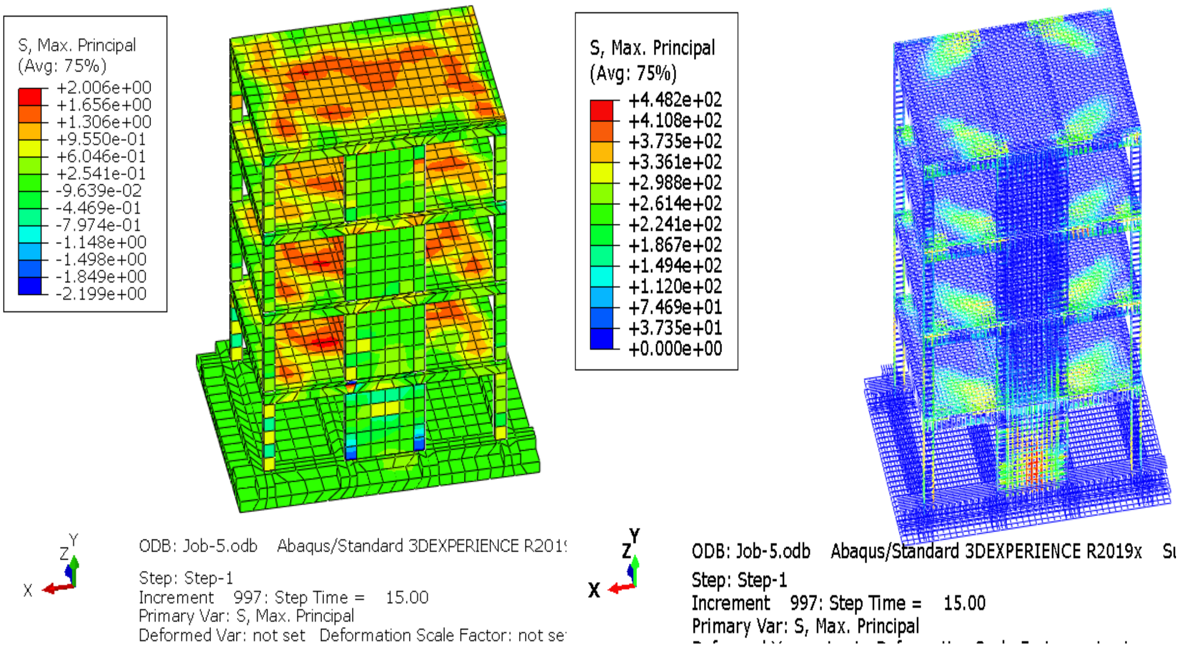


Figure 18. The max principal stresses in Concrete and rebars the model

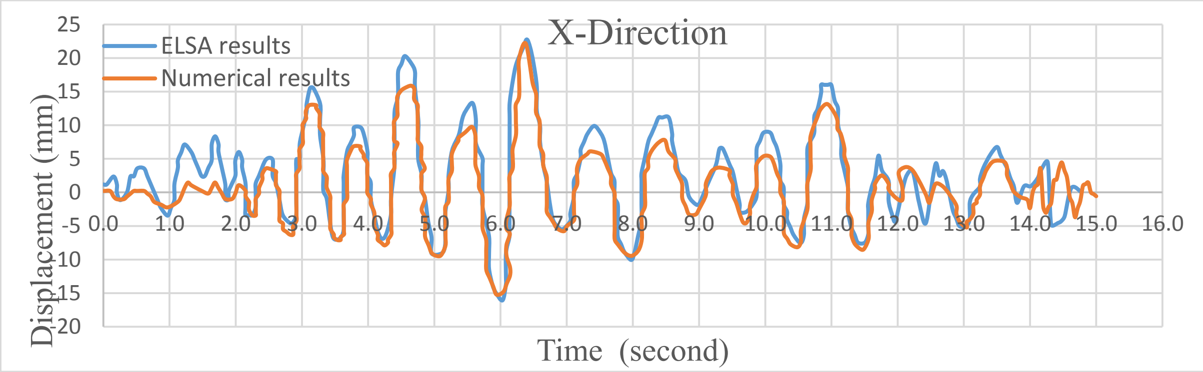


Figure 19. Experimental vs. Numerical results of ELSA, 0. 25g PGA testing and Abaqus result (in X-direction) for first floor.

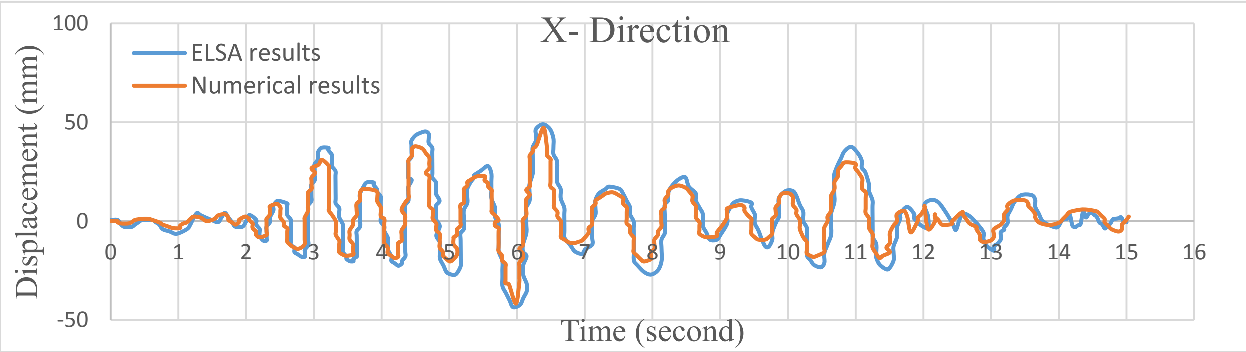


Figure 20. Experimental vs. Numerical results of ELSA, 0. 25g PGA testing and Abaqus result (In the X-direction) for second floor.

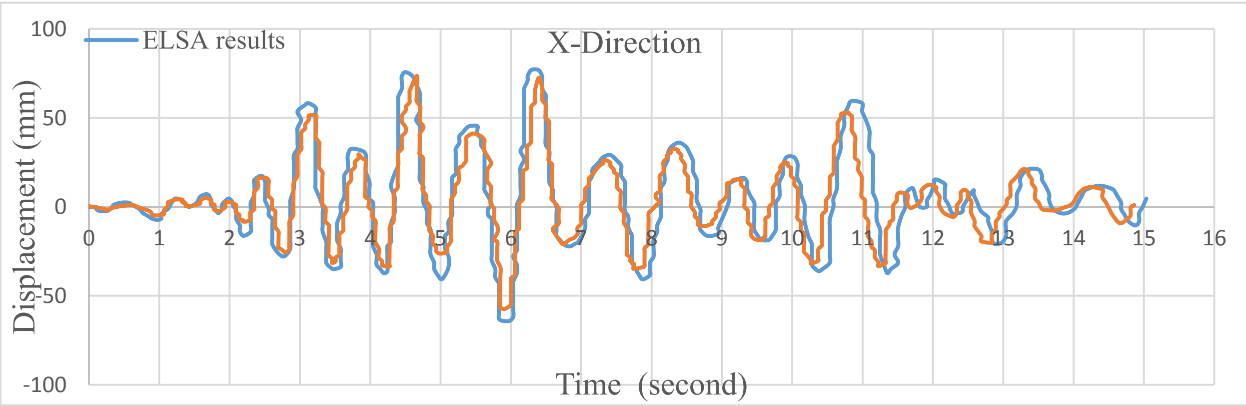


Figure 21. Experimental vs. Numerical results of ELSA, 0. 25g PGA testing and Abaqus result (In the X-direction) for third floor.

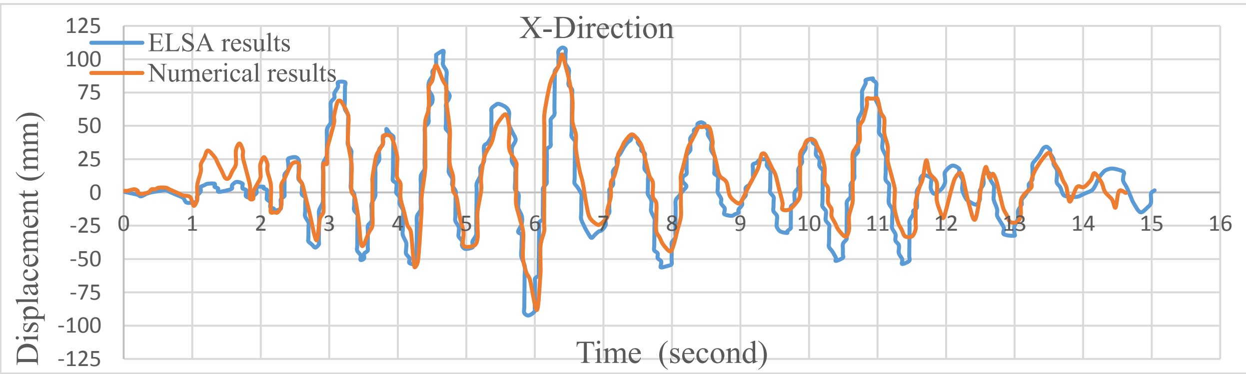


Figure 22. Experimental vs. Numerical results of ELSA, 0. 25g PGA testing and Abaqus result (In the X-direction) for Fourth floor

10. TABLES

TABLE I: Rebar properties based on data on material taken from ELSA laboratory

Bar dim. (mm)	Yield strength (MPa)	Yield strain	Ultimate strength (MPa)	Ultimate strain	Young's modulus	Poisson Coefficient
8.0	417.01	0.00226	583.68	0.132	206000	0.3
12.0	424.68	0.00222	570.32	0.173	206000	0.3
16.0	437.42	0.00213	546.69	0.141	206000	0.3
20.0	376.68	0.00182	567.32	0.167	206000	0.3

TABLE II: Concrete properties based on data on material taken from ELSA laboratory

Poisson Coefficient	0.2
Young's Modulus	30000 MPa
Tension Stress Limit	2.75 MPa
Tension Deformation Limit	0.00018
Stress Limit	25 MPa
Deformation Limit	-0.003

TABLE III: Structure model dimensions

Type of model		Dimension	Reinforcement(mm)
Beams	Transversal Beams	500*250mm	Top:2ϕ20, Bottom:5ϕ20 Stirrups: ϕ 10 @150
	Longitudinal Beams	500*250mm	Top:4ϕ12, Bottom:4ϕ12 Stirrups: ϕ 8 @200
Slabs		8900*6250*150 mm	ϕ10@150mm for top and bottom reinforcement
Columns		400*250mm	4ϕ20,Stirrups:ϕ8 @200
Foundation		11000*8000*400mm	ϕ16@250mm for top and bottom reinforcement
Tie Beams At Foundation		600*800*800	7ϕ16 for top rein, 4ϕ16 bottom rein and Stirrups:ϕ12 @175

TABLE IV: Reinforcement details and location of the RC infill walls in North Walls

Story	N Wall								S Wall							
	web bars	embedment of web starter bars, mm		Dowels					web bars	embedment of web starter bars, mm		dowels				
				Φ mm	embedment, mm							Φ mm	embedment, mm			
		in wall	in frame		bottom & east in:		top&west in:			in wall	In frame		bottom & east in:		top&west, in:	
					wall	frame	wall	frame					wall	frame	wall	frame
1	Φ12@200	600	230	Φ20	160	160	600	190	Φ10@200	500	170	Φ20	160	160	500	160
2	Φ10@200	500	170	Φ20	160	160	500	160	Φ 8@200	400	120	Φ18	145	145	400	145
3	Φ8@200			Φ18	400	145	400	145	Φ8@200			Φ16	400	130	400	130
4	Φ8@200			Φ16	400	130	400	130	Φ8@200			Φ16	400	130	400	130

TABLE V: Concrete damage plasticity Parameters

Dilation angle	34°
Viscosity	0.000050
Plastic strain ratio (Biaxial/Uniaxial compression)	1.160
Flow potential eccentricity	0.10
Invariant stress ration (Kc)	0.6670

TABLE VI: Comparison in Top Displacement between ABAQUS and ELSA test results

TYPE	Max Top Positive Displacement (mm)	Max Top Negative Displacement (mm)	The Difference between Top Positive Displacement	The Difference between Top Positive Displacement
Experimental results in (ELSA)	109	-93	2.47 %	3.12 %
Abaqus program results	106.3	-90.1		

11. CONCLUSION

The main validations result of this study are the following:

1-The building managed to sustain an earthquake of 0.25g without significant damage, this observed clearly in this paper by comparing the tested result gather experimentally at the Association's joint research Centre facility in Ispra (Italy) and the Abaqus (6.19) software analysis in which found out less than 4% difference between the data which can indicate a highly similarity between them.

2-The used by experimental tests can be time, resource consuming. However, by using numerical analysis specially FEA such as Abaqus software have the advantage of being ecumenical, time saving and more accurate.

3-Accordingly, this numerical modeling can be used for exploring other types of retrofitting systems such steel jacketing, Steel tube, base isolated and others strengthen teachings.

Reference

[1] E. Barkanov, "INTRODUCTION TO THE FINITE ELEMENT METHOD," Physics (2001).

[2] Abaqus, (2019), "Abaqus 6.19 Abaqus/CAE User's Manual".

[3] M.N. Fardis, A. Schetakis, and E. Strepelias, (2013), "RC buildings retrofitted by converting frame bays into RC walls", Bull. Earthq.Eng.,11(5),1541-1561.

[4] Ch. Z. Chrysostomou, M. Poljansek, N. Kyriakides, F. Taucer., F. J. Molina, "Pseudo Dynamic Tests on a Full-Scale Four-Story Reinforced Concrete Frame Seismically Retrofitted with Reinforced Concrete Infilling," Structural Engineering International, Volume 23, Number 2, pp. 159-166(8), May 2013.

[5] Ch.Z. Chrysostomou, N.C. Kyriakides, M. Poljansek, J. Molina, and F. Taucer, "Seismic Evaluation and Rehabilitation of Structures," Chapter 17: RC Infilling of existing RC structures for seismic retrofitting, Geotechnical, Geological, and Earthquake Engineering 26, Springer International Publishing, DOI 10.1007/978-3-319-00458-7_17, 2014.

[6] BS8110 (1983), Structural use of concrete, Code of Practice for Design and Construction, Part 1, The Council for Codes of Practice, British Standard Institution, London.

[7] CEN (2004), Euro code 2: Design of concrete Structures-Part 1: General rules for buildings (EN1992-1-1), Comet European Normalization.

[8] E. Georgiou, N. Kyriakides, Ch. Chrysostomou, P. Kotronis, P. Roussis, "Numerical simulation of the experimental results in an RC frame retrofitted with RC Infill walls," Earthquakes and Structures, Techno Press, 9 (4), pp.735-752, 2015.

[9] M. Poljansek, F. Taucer, J. Molina, Ch. Z. Chrysostomou, N. Kyriakides, T. Onoufriou, P. Roussis, P. Kotronis, A. Kosmopoulos Seismic Retrofitting of RC Frames with RC Infilling (SERFIN Project), JRC Scientific and Policy Report, Publications Office of the European Union, Luxembourg, doi: 10.2788/630.

[10] Infilling SERFIN Project, JRC Scientific and Policy Report, Publications Office of the European Union, Luxembourg, doi:10.2788/630, 2014.

[11] M. Turk, U. Ersoy, and G. Ozcebe, Retrofitting of Reinforced Concrete Frames with Reinforced Concrete Infills Walls. Proceedings of the fib Symposium on Concrete Structures in Seismic Regions, Athens, Greece, 2003.