Finite Element Simulation of the Device CAR1 on Braced Frames

***M.D. Titirla[1]**

[1]Department of Civil Engineering, Aristotle University of Thessaloniki, Greece.

*Presenting and corresponding author: mtitirla@civil.auth.gr

Abstract

The developed device, has the codename CAR1, belongs to the passive energy dissipation systems, as it doesn't require external power to generate system control forces. It can be used on new or existing structures and can be easily adapted to the particular demands of structures. It can be installed in a variety of ways such as in single or X diagonal bracing in building frames. Moreover the use of this device may result in improving (i) the increase of stiffness (ii) the absorption of seismic energy, (iii) as well as control of the axial forces that are developed at the diagonal steel braces. The main part of CAR1 device is the groups of superimposed blades, which absorb seismic energy through simultaneous friction and yield. Firstly this paper discusses the experimental and numerical evaluation of the effectiveness of this steel device. Full scale CAR1 device was experimentally investigated under cyclic loading in Laboratory for Strength of Materials and Structures of Aristotle University of Thessaloniki. Finite Element Models of CAR1 device were developed and analyzed using the software ABAQUS, checking the credible documentation of the device. In addition, a numerically robust finite element model of a whole one storey structure is described, for high-fidelity simulations of inelastic responses of device CAR1 on braced frame. Aim of this study is to compare the response of one storey structure with and without the existence of device CAR1 on diagonal braces.

Keywords: Experimental validation, Finite Element verification, Absorption Seismic Energy, Friction, Dynamic Explicit analysis.

Introduction

The safety of construction (existing or new) is one of the major priorities of engineering globally, because structures often subject to large and often devastating, for their viability, loadings. So, great interest is in the study of the innovations of the design and materials of construction that minimize the probability of failure of the structure in any charging.

Steel concentrically braced frames have been used widely in high-seismic regions due to their efficiency in meeting lateral-load resisting requirements. Based on extensive research since the 1970's, it is well known that the cyclic loading performance of steel braces depend on their slenderness ratio and on the width-to-thickness ratio of their cross sectional elements, and that adequate detailing of the bracing connection is critical to avoid premature fracture at the end of the brace. Braced frame systems are presently being designed to satisfy performance-based seismic design criteria [1, 2]. In terms of analysis capabilities, researchers have proposed methodologies to predict the occurrence of fracture from cumulative damage and to exhibit significant ductility in life safety and collapse prevention limit states, which are governed by inelastic post-buckling and tensile yielding behaviors of the brace elements.

There are essentially two main orientations in order to protect braced elements, first one is to provide the structural system in order to avoid unexpected premature failure modes (mid-

length or connections) and the second is to be incorporated in braces a passive energy dissipation devices.
Buckling restrained braced frames (BRBFs) for seismic load resistance have been widely used in recent years because it yields under both tension and compression without significant buckling [3, 4, 5]. Others researchers create numerical models to approach brace elements. Numerical models can be classified in three categories, the phenomenological models [6, 7, 8], the beam-column Finite elements models [9, 10, 11] and the 3-D Finite elements models [12, 13, 14, 15, 16, 17].
On the other hand, passive energy dissipation devices such as visco-elastic dampers, metallic dampers and friction dampers have widely been used to reduce the dynamic response of civil engineering structures subjected to seismic loads [18, 19, 20] and can easily replaced or repaired. Their effectiveness for seismic design of building structures is attributed to minimizing structural damages by absorbing the structural vibratory energy and by dissipating it through their inherent hysteresis behavior. So, several of these devices have been selected for seismic strengthening of existing or new buildings in the US, Canada and Japan [21, 22, 23].
In order to demonstrate the effectiveness of the devices, many passive energy dissipation systems were studied in experimental research [24, 25, 26, 27] or in numerical research [28, 29, 30]. The Finite Element Method (FEM) has become the most popular method in both research and industrial numerical simulations, as it takes into consideration material laws, contact interface conditions and others parameters, which lead to the exact response of the device. Several algorithms, with different computational costs, are implemented in the finite codes, such as ABAQUS [31], which is commonly used software for finite element analysis. Comparison of numerical results with the same experimental one is very useful and necessary as it provides the possibility to researchers to study the behavior of their devices more widely [32, 33]. The calibrated FEM models are used to conduct a series of simulations to study the effect of different parameters. In this way, results come out that are harder to obtain experimentally.
In the present paper, a numerically robust finite element model is described, which is based on explicit time-stepping, for high-fidelity simulations of inelastic responses of device CAR1 on braced frame. The effectiveness of the investigated device was recently developed at the Laboratory of Strength of Materials and Structures of Aristotle University of Thessaloniki. Aim of this study is to compare the response of one storey structure with and without the existence of device CAR1 on diagonal braces.

Study of the individual device CAR1

A short description:

The developed device has the codename CAR1 and belongs to the passive energy dissipation system, as it doesn’t require external power to generate system control forces. This device proposed by Papadopoulos et al. [34] and it consists of 4 main elements, as illustrated in Figure 1. Device CAR1 has the advantage to (i) provide additional stiffness as well as (ii) absorption of seismic energy, through yield and friction, (iii) provision of control of the axial forces that are developed at the diagonal steel rods and last but not least the ability to retain the plastic displacements to a desired level, due to the restrain bolt. Energy dissipation is provided by inelastic bending of superimposed blades.

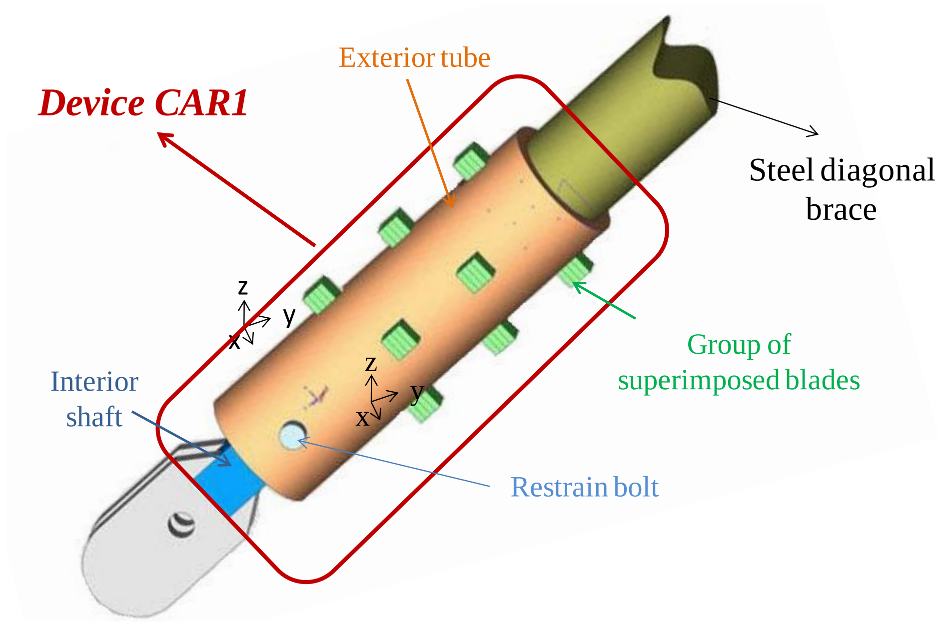


Figure 1: The investigated device CAR1.

Moreover, device CAR1 can be used on new or existing structures and can easily be adapted to the particular demands of structures. However, it can be installed in a variety of ways which include using them in single diagonal braces or in X braces (Figure 2) and in accordance with the requirements of each construction, it can be used one or more devices.

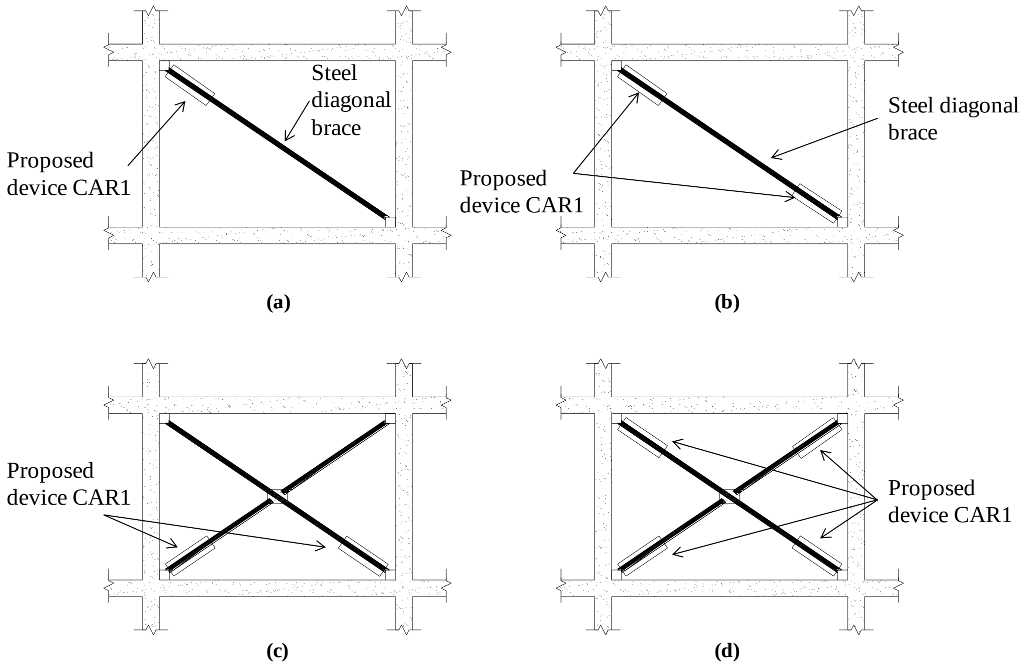
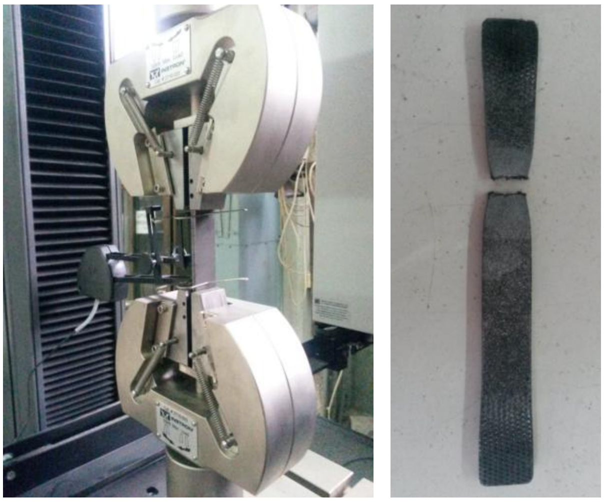


Figure 2: Possible positions of the device CAR1, incorporated in steel diagonal braces.

Experimental set up:

A standard test has been carried out in order to establish the basic material properties of the superimposed blades (Figure 3). These experimentally derived material properties were utilized in the subsequent numerical study.



Young Modulus E (Gpa)	203.6
Yield Strength F_y (Mpa)	220
Maximum Strength (MPa)	290

Figure 3: A standard test in order to establish the basic properties of the superimposed blades.

Full scale CAR1 device was experimentally investigated under cyclic loading. The experimental sequences have been conducted at the Laboratory of Strength of Materials and Structures of Aristotle University of Thessaloniki. The specimen details of the experiment are depicted in Figure 4. The load was controlled with a 100kN capacity load cell under deflection control. Two LVDT's were positioned at each side of the longitudinal axis of the device CAR1, which measure the relative movement of the interior shaft to the exterior tube. All data were recorded and were stored in a digital data system via a computer. We notice that only two group of superimposed blades were tested. Every group consists of five steel blades, each 4mm thick. Quasi-static cyclic tests were carried out in order to ascertain device's CAR1 behavior to absorbed seismic energy. The experimental sequence is 17 cycles displacement control with values starting from 4.5 mm up to 10 mm with rate 3mm/minute.

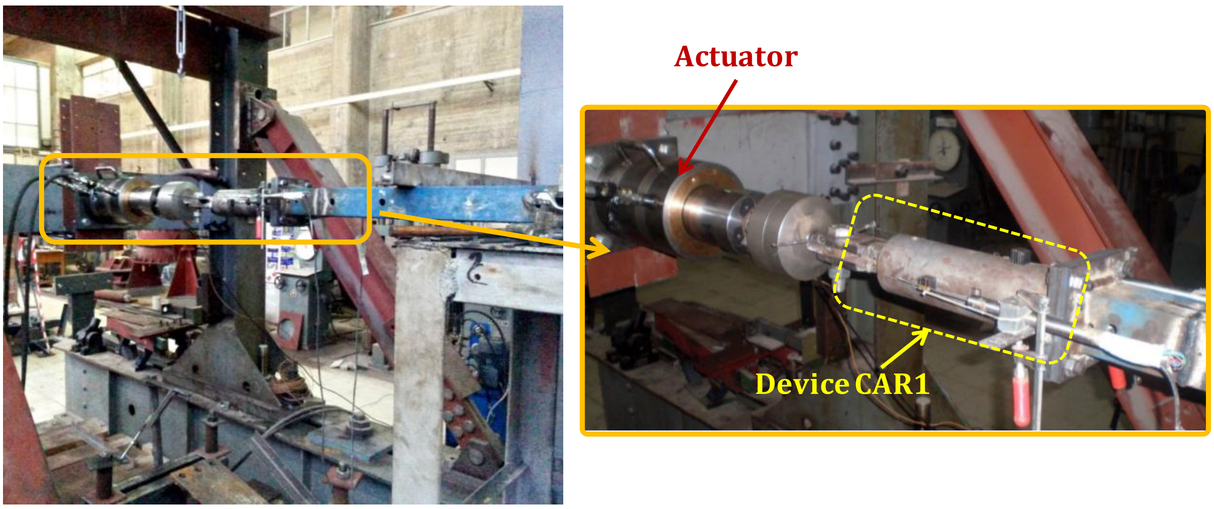


Figure 4: Specimen details.

Finite Element Modeling:

The general purpose FE software ABAQUS was employed to generate FE models to simulate numerically the behavior of the device CAR1. It was selected to use an explicit dynamic solver because this allows the definition of very general contact conditions for complicated contact problems, without generating numerical difficulties. The explicit dynamics analysis procedure is based upon the implementation of an explicit integration rule together with the use of diagonal ("lumped") element mass matrices.
To the comparison with the Standard, the explicit dynamic solver is computationally inefficient for quasi-static problems if real time is used, because the time needed to finish an analysis is proportional to its duration. However, it is often possible to scale the real time to a very small time period if the response of the structure remains basically static. According to classical dynamic theory, when a dynamic system is subjected to a linearly rising load, its response can be approximately treated as static if the duration of the loading stage is large compared to the natural period of the system. For solving this problem, check the ratio of kinetic to internal energy can be used to check if the structure has failed and the analysis is continuing simply as dynamic motion. It is stated in the ABAQUS/Explicit manual [31] that the procedure is quasi-static if the ratio of the kinetic energy to the internal energy is less than 2%. Any responses which have an energy ratio larger than this should be treated as dynamic and removed from the results.
The FEM model geometry reproduced the actual geometry of the tests set up of the device CAR1 to characterize the behavior of the device. The geometry of FE model was reproduced in full detail (Figure 5).

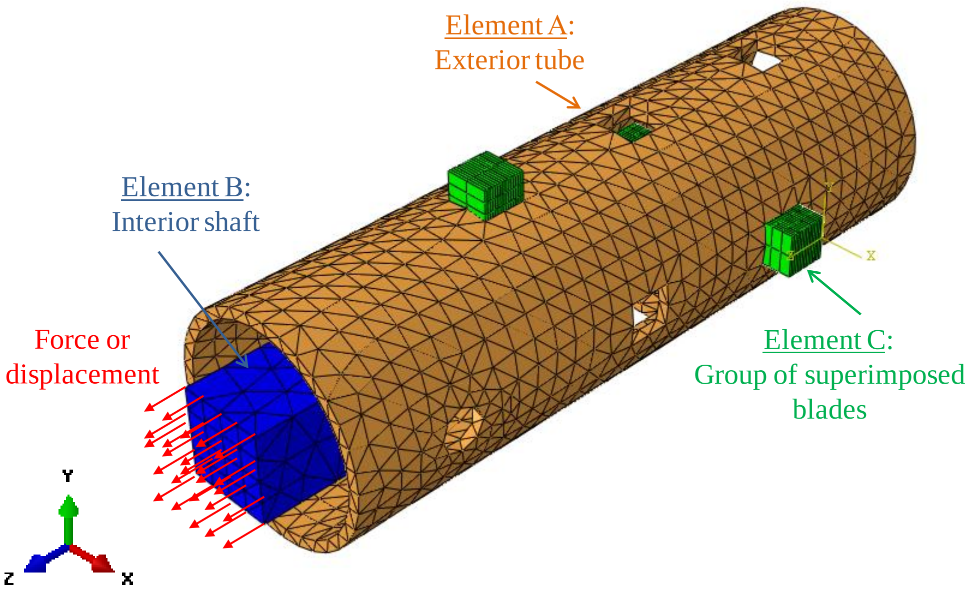


Figure 5: The FEM model used for the device CAR1 in software ABAQUS.

Several simulations were conducted to identify the best meshing. For the explicit method, blades and interior shaft are meshed using 3D reduced integration solid element **C3D8R** (eight-node bricks), while exterior tube is meshed using 3D solid element **C3D4** (four-node tetrahedron) available in ABAQUS. Normally, a higher mesh density provides for higher accuracy but also increases the computational time without improving substantially the accuracy of the results, therefore, a trade-off between time and accuracy becomes crucial [35].

The final mesh has 8126 elements and it resulted in a solution that correlated with the experimental results.
The uniaxial stress–strain relation of the blades, exterior tube and interior shaft are modeled as elastic with Young's modulus (Es) and Poisson's ratio (v) of which typical values are 200 GPa and 0.3, respectively. Plastic behavior are defined in a tabular form, including yield stress and corresponding plastic strain. The experimentally obtained stress (σ_{nom})-strain (ε_{nom}) curves for the blades was converted into the true stress (or Cauchy) (σtrue)-logarithic plastic strain (ε_{ln}^{pl}) format according to Eq. 1 and 2 and utilized to define the material response.

$$\sigma_{true} = \sigma_{eng}\,(1 + \varepsilon_{eng}) \tag{1}$$

$$\varepsilon_{pl} = \varepsilon_{true} - \frac{\sigma_{true}}{\mathrm{E}} \tag{2}$$

The surface-to-surface contact formulation technique with small sliding between the contacting surfaces was chosen. The contact definition includes the specification of two surfaces, one acting as the ''master'' surface and the other as the ''slave'' surface. The contact algorithm searches whether the nodes of the slave surface are in contact with the nodes of the master surface and enforces contact conditions in an average sense over a region of slave nodes using a Lagrange multiplier formulation [31]. A friction coefficient equal to 0.2 [36] was assumed between the contacting surfaces. A flowchart for carrying out the FEM analysis procedure is presented in Figure 6.

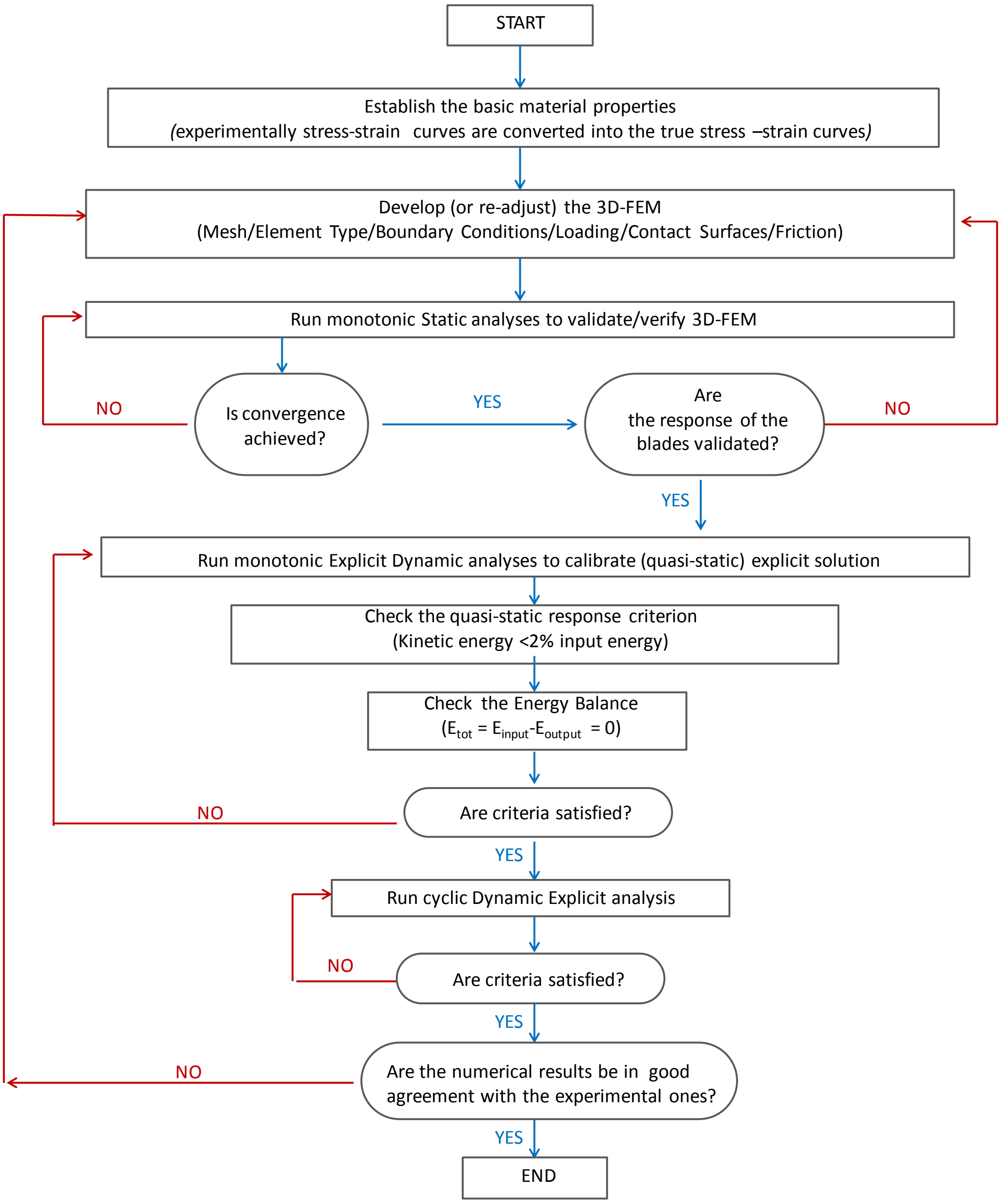


Figure 6: Flowchart in order to develop the Finite Element Model of Device CAR1 in ABAQUS.

Figure 7 plots the force versus relevant displacement from FEM analyses along with the experimental hysteresis. Blue lines illustrate hysteresis loops of experiments, while green lines shows hysteresis loops of Finite Element Models. The predicted values for the load and displacement are in very good agreement with the corresponding experimental ones. The comparisons between the FEM analyses and experiments show that the proposed FEM model is capable of reproducing the inelastic response of the device CAR1. Therefore, it is a reliable tool for the simulation of the hysteretic behavior of the device CAR1 and can be used to contact further studies to investigate the effect of various parameters. In addition, the area within a hysteresis loop is equivalent to the amount of seismic energy that the device is dissipating. Since the shape and consistency of the hysteresis loops, observe the device's ability to absorb CAR1 seismic energy, whereas will not break during the cyclic loading.

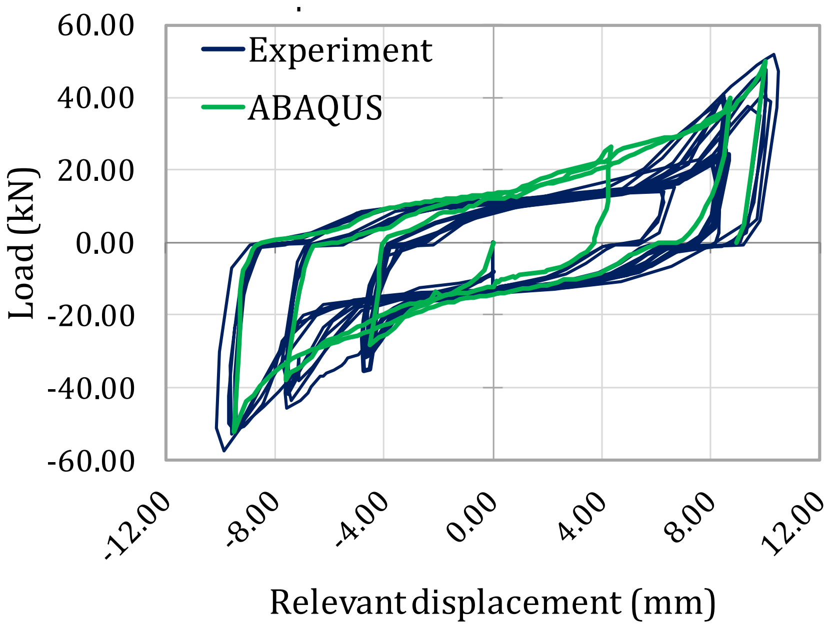


Figure 7: Comparison of the experimental and the numerical force–displacement hysteresis of the device CAR1

In addition, the numerical deformed shapes are compared with the corresponding experimental ones for relevant movement Uz=±5mm in Figure 8.

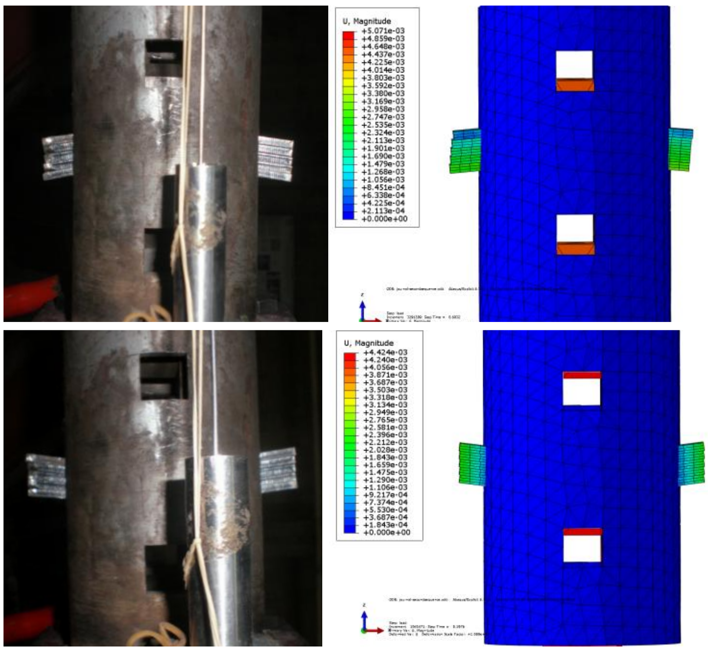


Figure 8: Distribution of deformed shapes (values in m)

Study of one storey structure with and without the existence of device CAR1on diagonal braces

A one storey reinforced concrete structure (Figure 9) was chosen to be studied. It has a height of 3m and length equal to 4.5m. The horizontal elements are beams with dimensions in plan 25x50cm and the vertical are columns with dimensions in plan 35x35cm.

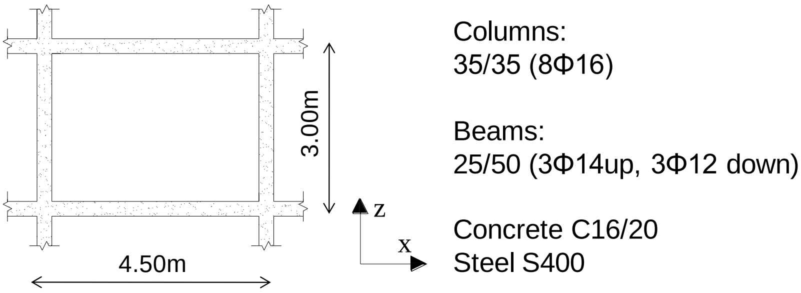


Figure 9: The longitudinal section of structure.

Structure was modeled and analyzed in SAP 2000 ver. 11.0.3 [37] in order to define floor's displacement drifts for seismic performance "Life Safety" (drift=1.6%) and "Collapse" (drift=2.1%). Columns and beams were modeled by frame elements. As it is drawn, the maximum horizontal displacement was chosen equal to 6cm (drift=2.0%), smaller than the collapse displacement (6.3cm). Also, the 2004 NEHRP provisions [38] allow the design of buildings with passive damping systems to experience controlled inelastic deformations associated with typical design drifts limits, e.g. a 2% drift limit. For this horizontal displacement, both braced structures ((i) with diagonal brace and (ii) with diagonal brace and CAR1 device) will compare in software ABAQUS.

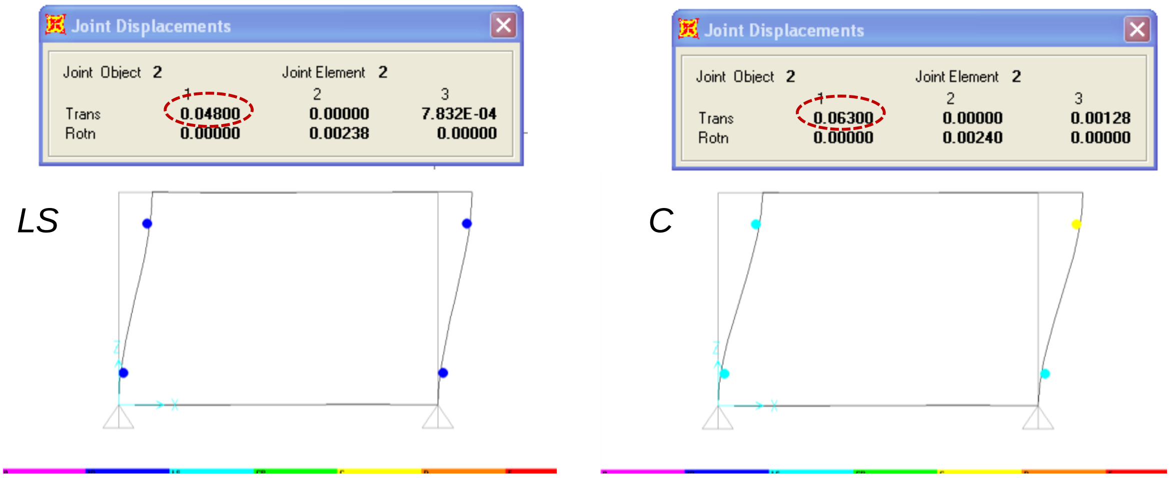
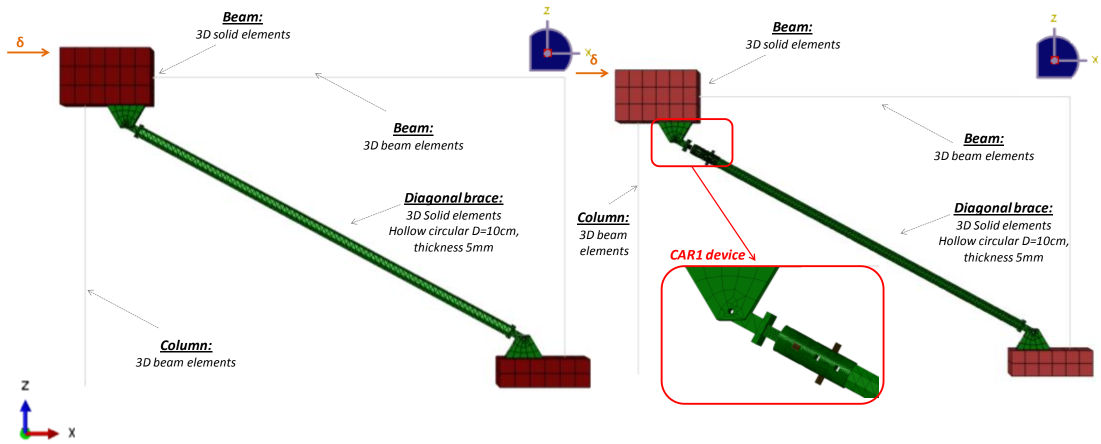


Figure 10: Deformed shape in SAP2000

Both braced structures were model and analyses in software ABAQUS, as it is illustrated in Figure 11. Columns were modeled with 3D beam elements while part of beam, diagonal brace and device CAR1 were modeled with 3D solid elements. The main parameters of modeling are mentioned in section 2.3 and it is not considered necessary to re-commented. Horizontal displacement (δ) imposed at the top of the floor increased step by step until the maximum displacement of 6cm. Dynamic Explicit analysis were contacted and useful results were observed.



(i) (ii)

Figure 11: (i) Braced Structure, (ii) CAR1-Braced Structure in ABAQUS.

Figure 12 shows the distribution of horizontal displacement around x-x axis at the end of analysis. In Braced Structure, diagonal brace fracture especially at the middle length of diagonal, while in CAR1-Braced Structure the brace remains un-deformed without plastic hinges. Figures 13 and 14 show the peak plastic strains at the end of the analysis. In braced structure, maximum plastic strain observed at the middle length of diagonal brace (fracture point), while in CAR1-Braced Structure at the superimposed blades, which are easily be replaced with minimum cost. As a result, using CAR1 device on diagonal brace, the fracture life of brace is increased. The system exhibited uniform energy absorption with more stability, as strength and maximum deformation of the system increased considerably.

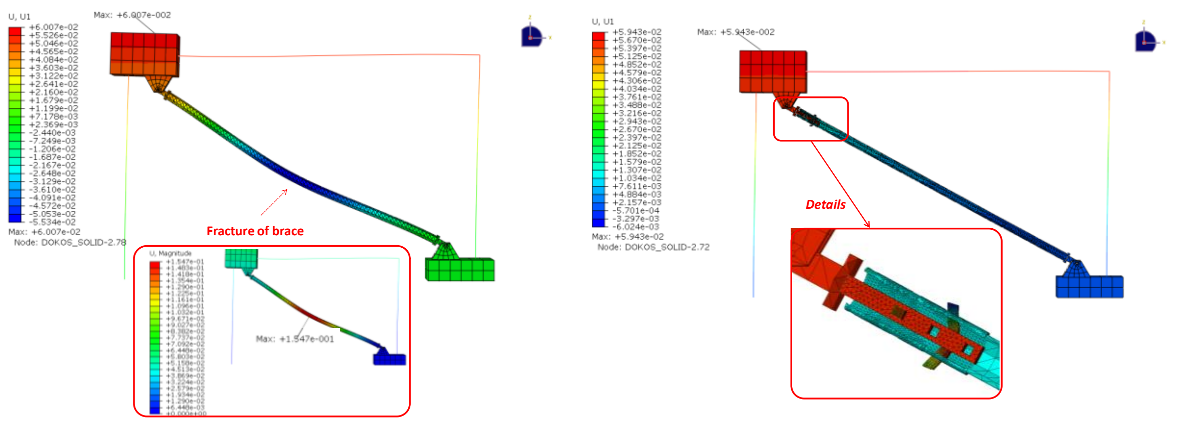


Figure 12: Deformed model at end of analysis, (i) Braced Structure, (ii) CAR1-Braced Structure (values in m).

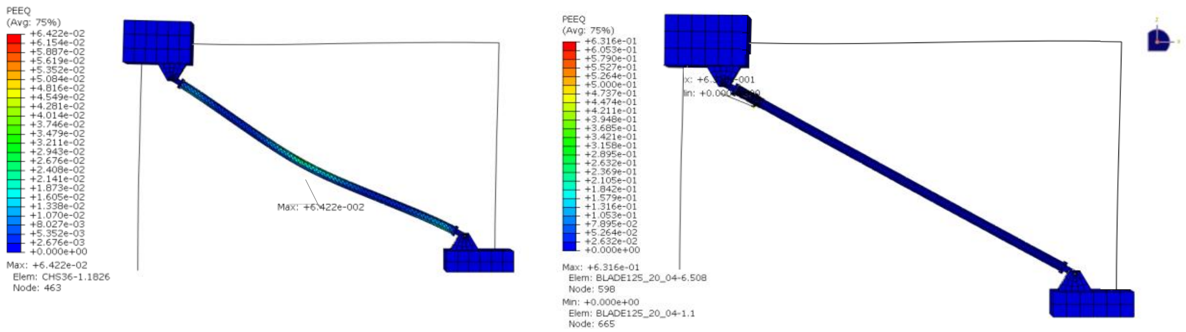


Figure 13: Deformed shape and maximum plastic strain at the end of analysis.

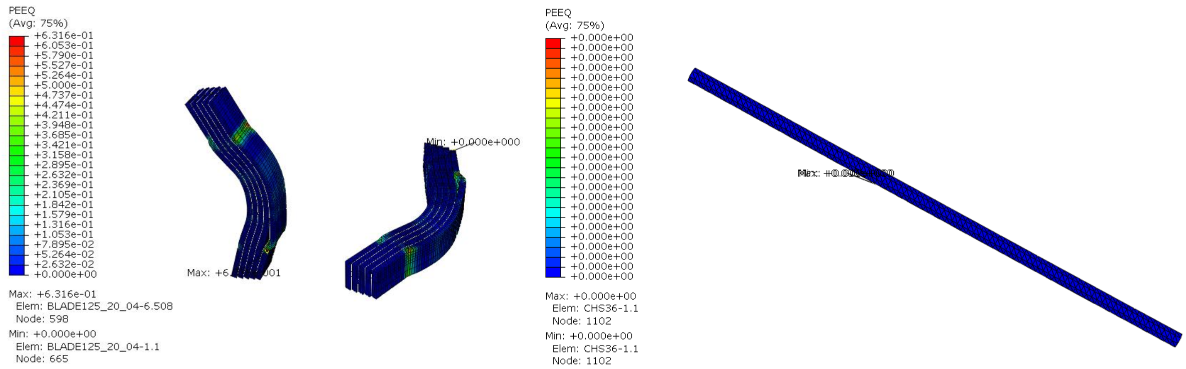


Figure 14: Maximum plastic strain in superimposed blades and diagonal brace at the end of analysis for CAR1-Braced Structure.

Conclusions

In the present paper, an anti-seismic steel device (with code name CAR1) for seismic strengthening of existing or new buildings, which was recently developed at the Laboratory of Strength of Materials and Structures of Aristotle University of Thessaloniki, is studied experimental. A detailed nonlinear finite element model (FEM) was also developed. This model was calibrated against experimental results and used to explain the response of the device CAR1. In addition, a numerically robust finite element model of a whole one storey structure is described, for high-fidelity simulations of inelastic responses of device CAR1 on braced frame.

Based on the findings of this paper, the following conclusions are drawn:

1. Device CAR1 is a reliable energy dissipated device, which can be used on new or existing structures and minimize the probability of failure of the structure in any charging.
2. The developed nonlinear FEM models can be reliable used to access the behavior of the proposed anti-seismic steel device CAR1 as they are capable to trace the hysteretic behavior and predict the deformed shape of the device with good accuracy.
3. Based on the shape and consistency of the hysteresis loops, it is recommended seismic energy, whereas will not break during repeated cyclic loading.

4. The calibrated FEM model permits a thorough investigation of the stress state in the blades and helps to identify all possible local failures
5. Using CAR1 device on diagonal brace, the fracture life of brace is increased. The system exhibited uniform energy absorption with more stability, as strength and maximum deformation of the system increased considerably.

Acknowledgments

I would like to sincerely thank my supervisor, Papadopoulo Paniko (Assistant Professor at Aristotle University of Thessaloniki), for his valuable support.

References

[1] Federal Emergency Management Agency (FEMA). FEMA 356: Pre-standard and commentary for the seismic rehabilitation of buildings, Washington, DC; 2000.
[2] American Institute of Steel Construction (AISC). AISC 341-10: Seismic provisions for structural steel buildings. Chicago, IL; 2010.
[3] Wada A, Huang YH, Iwata M. (1999) Passive damping technology for buildings in Japan. Prog Struct Engng Mater; **2(3**):1–15.
[4] Xie Q, (2005). State of arte of buckling-restrained braces in Asia, Journal of Constructional Steel Research, **61**, 727-748.
[5] Hoveidae, N and Rafezy, B.. (2012). Overall buckling behavior of all-steel buckling restrained braces. Journal of Constructional Steel Research **79**,151–158.
[6] Zayas VA, Popov EP, Mahin SA. Cyclic inelastic buckling of tubular steel braces. Earthq Eng Rsrch Ctr, Report No. UCB/EERC 80/16 1980; Univ. of California Berkeley, CA
[7] Ikeda K, Mahin SA, Dermitzakis SN. Phenomenological modeling of steel braces under cyclic loading. Earthq Eng Rsrch Ctr, Report No. UCB/EERC 84/09 1984; Univ. of California Berkeley, CA
[8] Khatib F, Mahin SA, Pister KS. Seismic behavior of concentrically braced steel frames. Earthq Eng Rsrch Ctr, Report No. UCB/EERC 88/01 1988; Univ. of California Berkeley, CA
[9] Ikeda K, Mahin SA. Cyclic response of steel braces. J Struct Eng ASCE 1986; **112(2)**:342-361.
[10] Jin J, El-Tawil S. Inelastic cyclic model for steel braces. J Eng Mech ASCE 2003; **129(5):** 548-557.
[11] Lee PS, Noh HC.(2010). Inelastic buckling behavior of steel members under reversed cyclic loading. Eng Struct. **32(9):** 2579–2595.
[12] Lotfollahi M, Alinia MM, Taciroglu E.(2011). Inelastic buckling simulation of steel braces through explicit dynamic analyses. Num Anal & Appl Math, ICNAAM, AIP Conf Proc 2011a; 1389: 2012-2015.
[13] Lotfollahi M, Alinia MM, Taciroglu E. (2011). A validated finite element procedure for buckling simulation of diagonally braced moment resisting frames. Num Anal & Appl Math, ICNAAM, AIP Conf Proc 2011b; 1389: 2016-2019.
[14] Lumpkin EJ, Hsiao PC, Roeder CW, Lehman DE, Tsai CY, Wu AC, Wei CY, Tsai KC. (2012). Investigation of the seismic response of three-story special concentrically braced frames. J Constr Steel Res; **77**: 131-144.
[15] Nip KH, Gardner L, Elghazouli AY.(2010). Cyclic testing and numerical modelling of carbon steel and stainless steel tubular bracing members. Eng Struct; **32(2):** 424-441.
[16] Yoo JH, Roeder CW, Lehman DE.(2008) Analytical performance simulation of special concentrically braced frames. J Struct Eng ASCE; **134(6)**: 881-889.
[17] Yoo JH, Roeder CW, Lehman DE. (2008). Influence of connection design parameters on the seismic performance of braced frames. J Constr Steel Res; **64(6):** 607-623.
[18] Aiken, I. D., Nims, D. K., & Kelly, J. M. (1992). Comparative study of four passive energy dissipation systems, Bulletin of the New Zealand National Society for Earthquake Engineering, **25(3)**, 175-192.
[19] Soong, T.T. & Jr, B.F.Spencer. (2002). Supplemental energy dissipation: state-of-the-art and state-of-the-practice. Engineering Structures, **24**, 243-259.
[20] Symans, M.D., Charney, F.A., Whittaker,A.S., Constantinou, M.C., Kircher, C.A., Johnson, M.W., and McNamara, R.J. 2008. Energy dissipation systems for seismic applications: Current practice and recent developments. J.Struct. Engrg., **134**(3-1).
[21] Pall, A.S., Verganelakis, V., March, C., (1987). Friction-Dampers for seismic control of Concordia University Library Building. Proc 5th Canadian Conference on Earthquake Engineering, Ottawa, pp 191-200.

[22] Makris, N. and Constantinou, M. C., (1992), .Spring-Viscous Damper Systems for Combined Seismic and Vibration Isolation,. Earthquake Engineering and Structural Dynamics, **21**, pp. 649-664.
[23] Martinez-Romero, E., (1993), .Experiences on the Use of Supplemental Energy Dissipators on Building Structures,. Earthquake Spectra, **9**, pp. 581-624.
[24] Whittaker, A. S., Bertero, V. V., Alonso, J. L. and Thompson, C. L. (1989). "Earthquake Simulator Testing of Steel Plate Added Damping and Stiffness Elements", Report No. UCB/EERC 89/02, University of California, Berkley.
[25] Anagnostides G., Hargreaves A.C., Wyatt T.A., (1989). Development and applications of energy absorption devices based on friction. J. Construct. Steel Research, 13, 317-336.
[26] Aiken, I. D., Nims, D. K., Whittaker, A. S., & Kelly, J. M. (1993). Testing of passive energy dissipation systems. Earthquake spectra, **9(3)**, 335-370.
[27] Papadopoulos, P.K., Salonikios, Th., Dimitrakis, S., Papadopoulos, A. (2013). Experimental investigation of a new steel friction device with link element foe seismic strengthening of structures. Structural Engineering & Mechanics, **46(4).**
[28] Pall, A.S., and March, C., (1982). Seismic response of friction damped braced frames. ASCE, Journal of Structural Division, **108 (9)**, 1313-1323.
[29] Papadopoulos, P. (2012). New nonlinear anti-seismic steel device for the increasing the seismic capacity of multi-storey reinforced concrete frames. The structural design of tall and special buildings, **21**, 750-763.
[30] Ramirez, J. D. M., & Tirca, L. (2012). Numerical Simulation and Design of Friction- Damped Steel Frame Structures damped. 15th World Conference in Earthquake Engineering.
[31] Abaqus Simulia, (2012). Analysis User's Manual Volume IV. Analysis User's Manual Volume IV . Providence: Dassault Systèmes.
[32] Vasdravellis, G., Karavasilis, Th., Uy, Br. (2013). Finite element models and cyclic behavior of self-centering steel post-tensioned connections with web hourglass pins. Engineering Structures, **52**, 1-16.
[33] Manos, G.C, Theofanous, M., Katakalos, K. (2014). Numerical simulation of the shear behaviour of reinforced concrete rectangular beam specimens with or without FRP-strip shear reinforcement. Advances in Engineering Software, **67**, 47-56.
[34] Papadopoulos P.K., Titirla M.D., & Papadopoulos A.P. (2014). A new seismic energy absorption device through simultaneously yield and friction used for the protection of structures. 2nd European Conference on Earthquake Engineering and Seismology, Istanbul.
[35] Doudoumis, I.N., (2007). Finite element modelling and investigation of the behaviour of elastic infilled frames under monotonic loading. Engineering Structures **29**, 1004–1024.
[36] Eurocode 3, (2003). Design of steel structures. Part 1-8: design of joints. prEN 1993-1-8:2003. European Committee for standardization: Brussels.
[37] Computers and Structures Inc (2007). SAP 2000 nonlinear version 11.0.3 User's Reference Manual. Berkeley, California
[38] BSSC (2004) NEHRP recommended provisions for seismic regulations for new buildings and other structures. Report FEMA 450, FEMA, Washington, DC