International Journal of Civil and Structural Engineering Research ISSN 2348-7607 (Online)
Vol. 6, Issue 1, pp: (213-221), Month: April - September 2018, Available at: www.researchpublish.com

Finite Element Analysis on Rocking Component of Cyclic Deformation of an Aseismic Reinforced Concrete Frame

[1]Soichiro Hirasawa, [2]Xiaozu Su

[1]Postgraduate Student, [2]Professor
[1,2]Department of Structural Engineering, Tongji University, Shanghai, China

Abstract: **During a strong earthquake, a reinforced concrete frame can have a non-linear behavior due to the cracking of the concrete on specific locations. This phenomenon is characterized for instance in columns, by a non-symmetric distribution of the vertical strain at the plastic hinge regions. This strain causes a certain elongation of the members, which enables the structure to have an elevation. This behavior could be assimilated to the rocking structures' one. Rocking structures are the structures, which can rock on their foundation, because they are not fixed to the ground. They have a good resistance against earthquake. In this paper, the elevation of a one-story reinforced concrete frame structure is studied. By using the finite element analysis software Abaqus, a pushover simulation is performed on a reinforced concrete frame, designed to be ductile according to the Eurocode. In the analysis, the global displacement of the top part of the frame is examined. Also in the columns, the vertical strain along the axis and the distribution of the vertical strain along the section are studied. As a result, it is shown that the frame has some significant elevation behavior, only when the structure yields. Also, during the pushover, the elongation of the columns does not decrease. This phenomenon could be explained by the important variation of the vertical strain in the plastic hinge of the columns.**

Keywords: **reinforced concrete frame structure, rocking structure, finite element analysis and pushover analysis.**

1. INTRODUCTION

During the last decades, rocking structures have been studied because of their good resistance against earthquakes. In fact, these structures, because they are not implanted in the ground, are able to rock on their foundation. Housner[1] is considered as the pioneer of this concept, and worked on the stability of a rocking rigid column. Different aspects, such as the behavior in dynamic loading have been developed. In most of the cases, the rigid hypothesis allowed a good understanding of some stone structures, like the ancient Greek temple, which underwent during the last two thousand years strong earthquakes. This observation is the starting point of Makris and Vassiliou's work[2][3][4]. But, the application of the rocking behavior has not been fully established yet and it is very limited for the use in ductile structures, like reinforced concrete structures, which represent the majority of the nowadays buildings.

Also, after the first simple rocking model from Housner, in 1993, Lipscombe and Pellegrinoz[5] intended to compare the numerical model of Housner and the experimental results. But the experimental results and the model did not match completely. They concluded that the behavior is complex and hard to be fully understood, especially for the phenomenon of the loss of the kinetic energy for each impact. Then, many scientists such as Peña et al[6][7] or Chin-Tung Cheng[8] tried to understand better the rocking behavior and tried to know more about it by doing experiments. We can say that the experimental studies give the real behavior of the different structures, and they enable to verify the theory and understand better the rocking blocks problematic. The numerical analysis and the theory have to share the same hypothesis, if not, the results cannot be compared. Also, concerning the experimental studies, one of the issues, was the need to face the difficulty of implementation. The cost of the experiment and the accuracy of the experimental devices could be discussed.

In civil engineering as well as other fields, the use of the computer programs has spread in the last two decades. Indeed, modeling enables to reflect real structure's behaviors quite accurately without conducting real experiments. The Finite Element Method (FEM) in particular, is often used to confirm or to obtain new results. Roh and Reinhorn[9] and Lu and Panagiotou[10] used this method to understand reinforced concrete structures with non-linear behavior, because it can provide response very similar to reality. In fact, in this research, the commercial software Abaqus[11] has been used for the different simulations.

Besides, the studies on the reinforced concrete frame from earthquake ground motion helps the well understanding on the ductility and the impact on the design of the structures. Clough et al[12] developed for instance the two-component model, where only the end of the element and in parallel insures the plastic behavior, the rest of the beam is linear elastic. Mahin and Bertero[13] have used this model to study the influence of the different ductility on the resistance of the structure against earthquake ground motion. Indeed, the ductility could be the key of the resistance of the reinforced concrete against strong ground motion. That is why some authors studied the impact of the size members on ductility. It is the case for Tawfik et al[14], who investigated by experimental process, if a relationship exists between global ductility and the size ratio for member for a one-story reinforced concrete frame. Also some studies, for instance Alebi and Kianosh[15] or Filiatrault et al[16], compared a normal design structure, that they called "nominally ductile" and a ductile structure, designed following the Canadian code[17].

During earthquake the different stresses induce some plastic hinges located in very précised place on the structure. Typically, these plastic hinges are located on the cross section between the beam and the column but also at the base of each column. These behaviors are modeled normally as a plastic rotation structure and then used for the calculation of the strength of the frame. But it can be supposed that these non-elastic behavior models are not accurate enough to understand fully the behavior of the reinforced concrete frame structure. Indeed, it can be assumed that during the loading, the center of gravity can lift because of the vertical strains. Thanks to this rocking behavior, a new component is added, because of the elevation of the center of gravity. It can be believed that the lifting in a classical reinforced concrete structure actually exists, which phenomenon generates a component, which makes the structure resist against the earthquakes. In this paper, this component will be defined as the rocking component. The study investigates on the impact of the rocking component on the behavior of reinforced concrete frame during an earthquake. Also, the influence of the rocking component on the behavior of the structures is studied.

2. DESIGN AND MODEL

2.1 MATERIAL:

There are two materials to be defined in our project, the concrete and the steel for rebar. The Eurocode suggests different choices for these materials, and for this topic, the concrete C25/30 and the steel C450 FE500 are chosen. This part gives the main information of these materials, and how they have been settled in the model is explained.

For the concrete, these following hypothesis have been settled: Compressive strength f_{ck} = 25 MPa, tensile strength f_{ctm} = 2.6 MPa, modulus of elasticity E = 31 GPa, specific weight density 25 kN/m^3, Poisson's ratio v = 0.2, Poisson's ratio in cracking area v = 0, factor of thermal expansion $A = 10^{-5}$ °C^{-1}, and deformation ε_{c2} = 0.002.

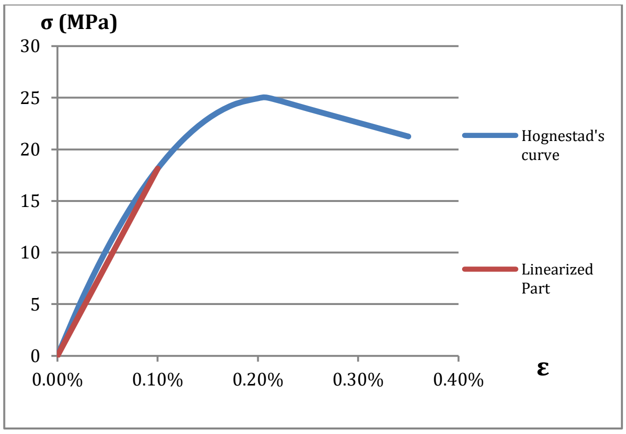


Fig 1: Stress-strain curve of the concrete in compression for the model

It was assumed that the concrete has an elastic behavior until a specific value. After this value is reached, the concrete follows the Hognestad curve[18]. Fig. 1 shows the linear curve in red, passing by one of the point (0.001; 18.4 MPa) of the stress-strain curve. Then, it was assumed that until this point, the behavior of the concrete is linear, and after this, the behavior becomes plastic and follows the Hognestad curve. For the tensile behavior, it has been assumed that the concrete has a linear behavior until its tensile strenght reaches the limit of f_{ctm}. This hypothesis is accepted after the pre-simulation done on Abaqus, which gives a good compromise between the results and the time for getting it on computer.

The rebar steel C450 FE500 is a highly ductile steel. In the design, the steel has the following characteristics: Plasticity limit f_y= 500 MPa, compressive strength f_t= 575 MPa, modulus of elasticity E = 200 GPa, specific weight density 78 kN/ m^3, ultimate deformation ε_{uk} = 0.075.

For general engineering applications, an elastic-plastic relationship is normally assumed. In an elastic hardening model, it is shown that the reinforcing steel has some hardening after it yields, according to Supaviriakit et al[19]. An elastic perfectly plastic model is generally acceptable for response prediction in reinforced concrete study according to Neale et al[20]. In Eurocode 2, for reinforcing rebar steel, this model is used.

2.2 THE STRUCTURE:

Because the goal of this paper is to study the rocking behavior of the concrete frame structures, the choice of the structure to model is very important. The location and the function of the building is adapted to the topic and is designed as close to the reality as possible. From this building, a typical frame is isolated to be modeled and studied by Abaqus computer simulation.

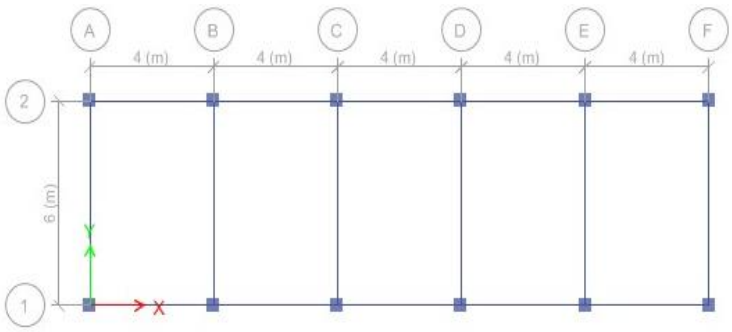


Fig 2: Studied frame structure from Etabs 2D display

In this part, the characteristics of the building and the reason of these choices are warranted. Then for the design, it follows the Eurocode[21]. To carry out the design, the commercial software Etabs[22] has been used. The predimensioning is based on Provost and Delpire[23] and Husson[24] handbook. They insure that the norms inside follow the different rules and properties of reinforced concrete from the Eurocode 2.

For the beams in X direction: Span L_x = 4 m, depth $h_x = L_x /10$ = 40 cm, and width $b_x \leq h_x$, b_x = 40 cm. For the beams in Y direction: Span L_y = 6 m, depth $h_y = L_y /10$ = 60 cm, and width $b_y \leq h_y$, b_y = 40 cm.

For the columns, it has been assumed that every column has the same design, whatever the location in the building. The principal problem in column is the buckling. To avoid it, it is suggested in the handbooks that for square column section $a \geq 0.7L/7.25$ = 38.8 cm (with L, the height).

For the structural part of the roof, the handbook gives the following requirement for the thickness of the roof slab: $e \geq L_y /30$ = 13.3 cm, with L_y the biggest dimension of the slab and e = 14 cm has been chosen.

To simplify the design, the reinforcement of the slab is not calculated and it is assumed that the frame structure (beams and columns) insures the mechanical stability of the building. Thus, the dead load includes the roof weight and the associated loads, such as the weight of the ceiling, the ventilation system, the ceiling-mounted lights and partitions. These permanent charges, according to the handbook[24], are equal to 2 kPa, all included.

Then, the dead load is as following: Dead load = roof weight + permanent charges = 5.4 + 2 kPa = 7.4 kPa. For the live load, the Eurocode 1 [25] gives the imposed loads for roofs with less than 5° slope as equal to: Live load = 1.5 kPa.

For earthquake load, the following parameter has been settled: Ground acceleration equal to 0.45 g, spectrum type 1, ground type A, the behavior factor equal to 5, the correction factor lambda equal to 1.3 and the eccentricity ratio equal to 0.05. All the parameter are from Eurocode 8. For the design mass, it also follows the code's requirement.

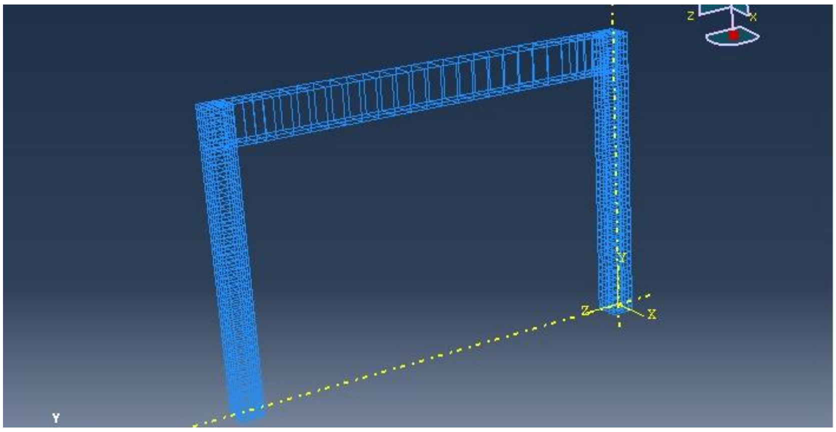


Fig 3: Rebar cage from Abaqus model

The rebar design is given by the following. In the columns: section of 12 longitudinal rebar of 14 mm of diameter with ties of 10 mm of diameter, spaced each of other from 100 mm. In the beam (*Y* direction), section of 4 longitudinal rebar of 20 mm of diameter, with 3 stirrups on the side, of 8 mm of diameter, spaced of 200 mm and in the center, 18 stirrups of 8 mm of diameter, spaced of 250 mm. The rebar cage is shown in Fig. 3. The concrete cover of 35 mm is taken.

3. PUSHOVER SIMULATION AND RESULT

3.1 SIMULATION STATEMENT:

Before beginning the simulation, an adapted statement has to be settled to carry out the pushover analysis on Abaqus.

For the meshing, Abaqus suggests three different geometries for the element types: Hex, Wedge and Tet. For this paper, a linear order of Hex (cubic element) has been chosen (C3D8), with reduced integration. In fact the choices of the meshing have been changed many times during the simulation before to settle the final meshing. So the average final size of one element of the concrete frame was 0.08 m for the width. For the rebar, close to those sizes have been settled for each element

During the design in Etabs, the Live and Dead loads on the beams have been defined, but for the model in Abaqus, the width of the beam has to be considered. Then the linear loads have to be divided by the width (=40 cm). Then the new loads on the Abaqus structure are determined accordingly. Also it has been assumed that all the structure undergoes the gravity namely an acceleration equal to -9.81 $m.s^{-2}$ for the vertical component.

For the boundary condition, the structure is considered as built-in the ground.

For the pushover simulation, there is the problem of the location and the type of the lateral load. Indeed, in a real experiment, a certain lateral pressure is applied on the top part of the frame, in axis of the beam. The decision was to apply the load in one point, in the lateral surface of the concrete frame, in the axis of the beam. But to avoid the local failure of the concrete, a steel cushion has been added on the model. Because the cushion needs to have a high elastic modulus, the material has been taken as the same as the rebar steel.

3.2 PUSHOVER LOADING:

Then for the pushover analysis, the lateral loads are increasing at each step. For the simulation in this topic, each time, the load has been increased by 30 kN (except for the last one). Thus, Fig. 4 gives the shape of the lateral load, function of the step number. For the intensity of the load, the sign "+" is used for the pulling and the sign "-", for the pushing. The first step corresponds to the initial step, and the second one to the step where only the dead and live loads are applied. In total, there are 49 steps, and it was assumed that this number is acceptable because of the limitation of the time and the computer capacity. Also for each 30 kN, a "check point step" is applied by increasing or decreasing the load profile. These steps allow studying in detail the behavior of the frame during the simulation and not only for the extreme value's loads.

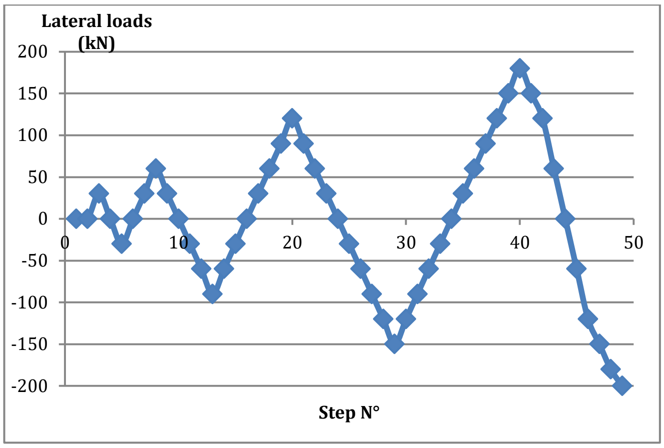


Fig 4: Lateral loads for each step

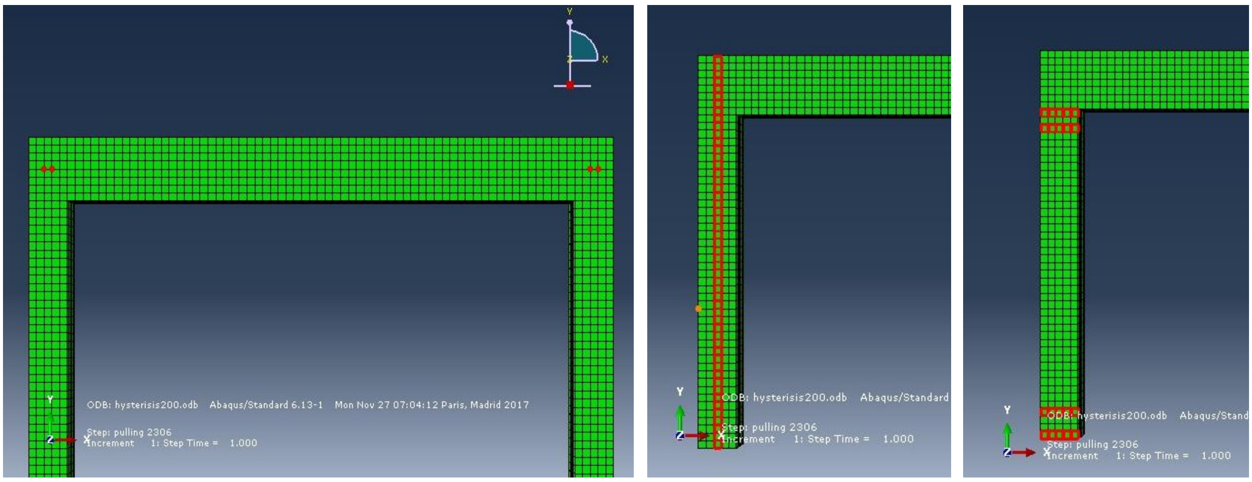
3.3 THE OUTPUT:

It has been assumed that the behavior of the frame is only in 2 dimensions, so there is no need to dig into the structure to collect information. Also, the analysis was in fact done for the two columns but it was chosen in this project to only show the left one because of the similarity of the results.

Displacement point: as it can be seen on Fig. 5a, the 4 single nodes (in red) were selected to study the general displacement of the frame. Indeed, for each cross section, there are 2 nodes to understand the behavior of the intersection point of the axis of the beam and the axis of the column (center of the cross section). U1 and U2 are defined as the average value of, respectively the lateral and vertical displacement of these four points. U1 and U2 are taking positive respectively in the direction of *Y* and *X*.

Vertical stain of the column: as it can be seen in Fig. 5b, all the element of the axis of the column has been selected. The elements are numbered from 1 to 51, where the element 1 is on the top of the column and the element 51 is on the bottom. For this study, the value of the vertical strain has been taken for the integration point of the element.

Vertical strain distribution in a column section: as it can be seen in Fig. 5c, the element of some specific section has been selected. These sections have the particularity to have the highest vertical strains among the column. For each section, the element is numbered from 1 to 5, where 1 is the element on the extreme left and 5 is the extreme right one. The value of the vertical strain has been taken to the integration point of the element.

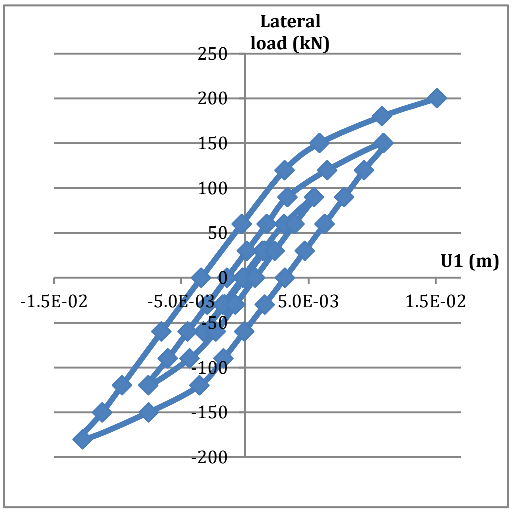


(a) **(b)** **(c)**

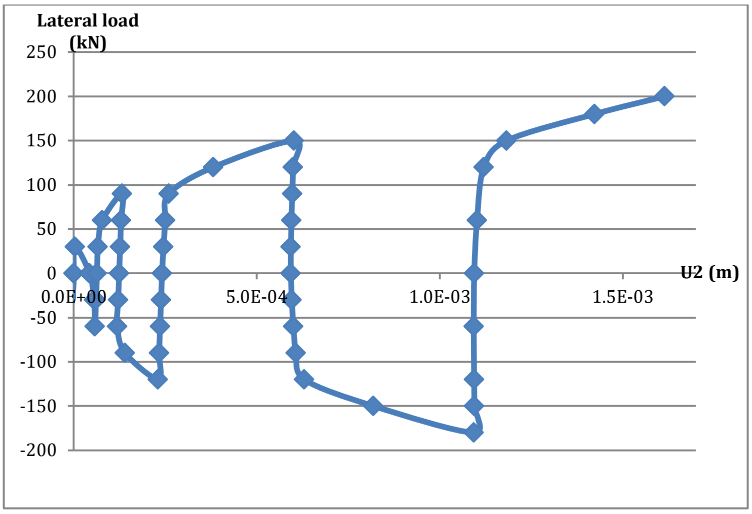
Fig 5: The location of the point and element for the output in the simulation

3.4 DISPLACEMENT OF THE FRAME:

In Fig. 6a, it could be seen the hysteresis behavior of the concrete frame, and that it corresponds to the elasto-plastic model. The behavior of the frame is linear elastic for a low value of the lateral load and enters in the non-linear plastic behavior. Indeed, the structure does not come back to its initial position.



(a)



(b)

Fig 6: The lateral and vertical displacement function of the lateral load

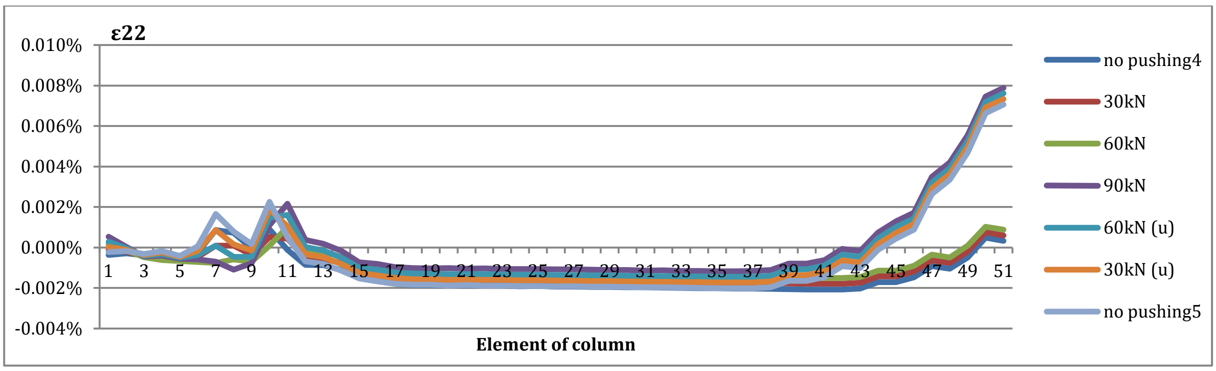
The Fig. 6b is the key result of this research; it shows the vertical displacement of the top part of the reinforced concrete frame, during the simulation. The first observation is that the vertical displacement almost never decreases and only increases. It could be explained by the fact that the variation of the vertical strain behavior is different during the elastic and plastic behavior. In correlation with Fig. 6a, only during the yielding phase the structure has some significant lifting. Thus during the unloading, the variation of the vertical displacement is almost non-existent. Also for each loop, the value of the lateral loads for which the structure begins to yield increases. That means that the value of the lateral loads for which the structure begins to lift increases for each loop.

3.5 VERTICAL STRAINS ALONG COLUMN AXIS:

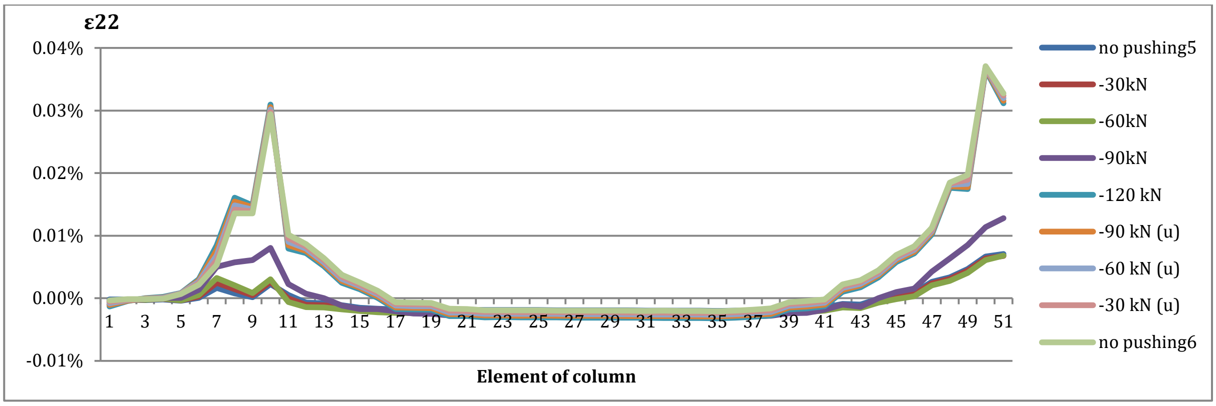
The graphs in Fig. 7 show the vertical strain along the axis of the left column, for the 90 kN and -120 kN loading and unloading phase.

It could be seen in both graphs that, especially in the bottom of the column, the vertical strain ($e22$) does not have a significant variation until a certain value of lateral load (60 kN for Fig. 7a and -60 kN for Fig. 7b), but after a new loading, the variation is very important. This variation is made only on the very precise locations in the column: in the bottom and near the cross section between the beam and column. During the unloading (with a "(u)" in the caption), for both curves, the variation of the strain is non-existent, and the integration point keeps its value of the strain. In fact by linking the data from the two graphs, even during the loading phase of Fig. 7b, the variation of the strain is not significant until the yielding.

Also for Fig. 7a, it seems that a plastic hinge is created in the bottom of the frame. In Fig. 7b, this hinge becomes bigger and a new hinge, near the cross section is created in the column. Also by comparing the graphs, the maximum value of the vertical strain increased of 500%. In fact, after the tensile failure of a concrete element, only the steel rebar undergoes the stress. Thus, this quick variation could be explained by the high reduction of flexural stiffness of the section.



(a) For 90 kN loading-unloading



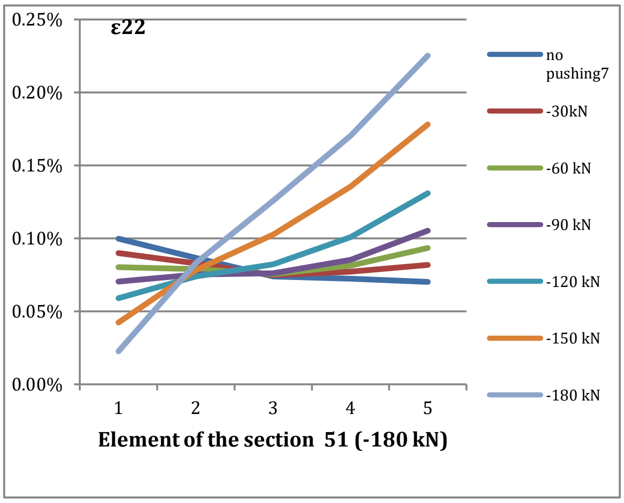
(b) For -120 kN loading-unloading

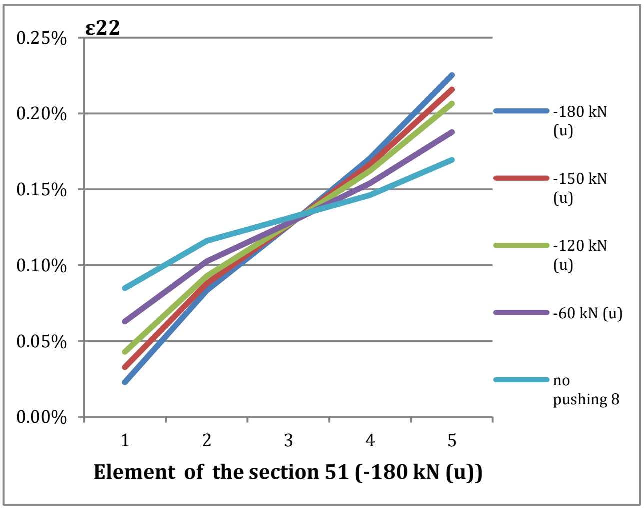
Fig 7: Vertical strain distribution along column axis

3.6 VERTICAL STRAINS ALONG CRITICAL COLUMN SECTION:

To understand more in detail the previous phenomena, the vertical strain in the section of column has been studied (Fig. 8). Some specific graphs have been selected to underline and explain the previous results. The graphs from Fig. 8 show the vertical strain in the section 51, the section on the bottom of the column, for the -180 kN for loading and unloading phases.

In the -180 kN loading curve (Fig. 8a), all the concrete is in positive strain. It could be observed that for the element 2-3-4, the variation of the strain is limited until the yielding. But in plastic behavior the vertical strain profile becomes increasingly non-symmetric, showing serious lifting effect. In the -180 kN unloading curve (right), the decrease and increase of the strain are more symmetric and centered in the middle element, where there is no variation. For the other elements, the varying seemed to be linear in unloading.





(a) For the -180 kN loading **(b) For the -180 kN unloading**

Fig 8: Vertical strains along the column bottom section (section 51)

4. CONCLUSION

As conclusion, it can be said that the pushover analysis showed that there is lifting of the RC structure when the structure yields. Indeed, in plastic behavior, the vertical strain in the columns has important variations, which allow the column to have a significant elongation. It is due to the fact that in tension, only the reinforced rebar works, and creates on asymmetric distribution of the vertical strain in the section of the column. So it can be assumed that during a strong ground motion, when the structure yields, there is the existence of the rocking component, which helps the structure to resist against the earthquake. Also, it can be imagined that the more the reinforced concrete frame has a high ductile behavior, the more obvious will be the rocking component.

Also it has to be indicated that even there is a significant lifting, there is no decrease of the elongation of the element and the structure only get higher. First of all, only a static analysis have been made in this research, so it would be interesting to do dynamic simulations to see if the results of these simulations are similar or if the rocking behavior of the frame exists also when earthquake load is applied on the structure. The last but not the least, in the design provided by Etabs, the beam was too strong and did not yield as it was expected. Indeed, a weak beam strong columns design would be better to understand the problem and give a more realistic approach. Also, if the slab was designed in Etabs and added in the model on Abaqus, the horizontal member, composed by the beam and the slab, will be stronger and may affect the result.

REFERENCES

[1] Housner, G. W., "The behavior of inverted pendulum structures during earthquakes", Bull. Seismological Soc. Amer., 1963, 53(2), 404-417.

[2] Makris, N., and Vassiliou, M. F. "Planar rocking response and stability analysis of an array of free-standing columns capped with a freely supported rigid beam", Earthquake Engineering and Structural Dynamics. 2013; 42(3), 431-449.

[3] Makris, N. and Vassiliou, M. F., "Are some top-heavy structures more stable?" J. Struct. Eng., 2014, 140. Doi: 10.1061/(ASCE)ST.1943-541X.0000933

[4] Makris, N. and Vassiliou, M. F., "The dynamics of the rocking frame", Seismic Assessment, Behavior and Retrofit of Heritage Buildings and Monuments, Computational Methods in Applied Sciences 37, 2015, DOI 10.1007/978-3-319-16130-3_2

[5] Lipscombe, P. R. and Pellegrinoz, S., "Free rocking of prismatic blocks", J. Eng. Mech. 1993, 119(7), 1387-1410.

[6] Pena, F., Prieto, F., Lourenço, P.B., Campos Costa, A., Lemos, J.V., "On the dynamics of rocking motion of single rigid-block structures", Earthquake Engineering and Structural Dynamics. 2007, 36, 2383–2399.

[7] Pena, F., Prieto, F., Lourenço, P.B., Campos Costa, A., "Experimental dynamic behavior of free-standing multi-block structures under seismic loadings", J. Earthq. Eng., 2008, 12, 953-979.

[8] Cheng , C. T., "Energy dissipation in rocking bridge piers under free vibration tests", Earthquake Engineering and Structural Dynamics, 2007, 36(4), 503-518.

[9] Roh, H., and Reinhorn, A. M., "Modeling and seismic response of structures with concrete rocking columns and viscous dampers". Eng. Struct. 2010, 32:2096-2107.

[10] Lu, Y., and Panagiotou, M., "Earthquake damage-resistant tall buildings at near fault regions using base isolation and rocking core walls", Conference: Structures Congress 2015, DOI: 10.1061/9780784479117.108

[11] Simulia, "Abaqus unified CAE, complete solutions for realistic simulation". [Online] https://www.3ds.com/products-services/simulia/products/abaqus/ [Retrieved in 11/2017].

[12] Clough, R.W., Benuska, K.L., and Wilson, E.L., "Inelastic earthquake response of tall buildings", Proceedings, Third World Conference on Earthquake Engineering, New Zealand, Vol. 11, 1965, New Zealand National Committee on Earthquake Engineering.

[13] Mahin, S.A. and Bertero, V.V., "Problems in establishing and predicting ductility in seismic design", International Symposium on Earthquake Structural Engineering, St. Louis, Missouri, USA, 1976.

[14] Tawfik, A. S., Badr, M. R., and ElZanaty, A., "Behavior and ductility of high strength reinforced concrete frames", HBRC Journal, 2014, 10:215-221.

[15] Alebi, S. T., Kianoush, M. R., "Behavior of reinforced concrete frame designed for different levels of ductility", 13th World Conference on Earthquake Engin., Vancouver, B.C., Canada, August 1-6, 2005, Paper No. 505.

[16] Filiatrault, A., Lachapelle, E., and Lamontagne, P., "Seismic performance of ductile and normally ductile reinforced concrete moment resisting frames", Experimental Study, Canadian Journal of Civil Engineering, 1998, 25(1).

[17] CAN3.A23.3.1994. "Design of Concrete Structures" Canadian Standards Association, Rexdale, Ontario, Canada, 1994.

[18] Hognestad, E., Hanson, N. W., and McHenry, D., "Concrete stress distribution in ultimate strength design", Journal Proceedings, 1955, 52(12):455-480.

[19] Supaviriyakit, T., Pornpongsaroj, P., and Pimanamas, A., "Finite Element Analysis of FRP Strengthened RC Beam", Songklanakarin J. Sci. Technol. 2004, 26(4): 497-507.

[20] Neale, K., Ebead, U., Abdel Baky, H., Elsayed, W., and Godat, A., "Modeling of deboning phenomena in FRP-strengthened concrete beams and slabs". Proceedings of the international symposium on bond behavior of FRP in structures (BBFS), 2005.

[21] Eurocodes, EN 1998: Design of the structures for earthquake resistance. [Online], http://eurocodes.jrc.ec.europa.eu/showpage.php?id=138, [Retrieved in 10/2017]

[22] Computers and Structures. Inc, Etabs 2016, [Online] https://www.csiamerica.com/products/etabs [Retrieved in 12/2017]

[23] Provost, M., Delpire, D., "Manuel de prédimensionnement des éléments structures des ossatures en acier, bois et béton armé". Version 3 – 8 septembre 2013.

[24] Husson, J. M., "Etudes des structures béton aux Eurocodes", Casteilla, 2012,

[25] Eurocode 1, EN 1991-1-1, "Section 6, Charges d'exploitation des bâtiments. 6.3.4 Toitures." 2002.