FBU-DAE 2023
3 (2) : 169-177
DOI: 10.59732/dae.1284381

IMPACT EFFECT ON THE EXISTING AND STRENGTHENED REINFORCED CONCRETE COLUMNS

MEVCUT VE GÜÇLENDİRİLMİŞ BETONARME KOLONLARDA ÇARPMA ETKİSİ

R. Tuğrul ERDEM*, Engin GÜCÜYEN**

Geliş Tarihi/Received: 17 Nisan 2023
Kabul Tarihi/Accepted: 7 Ekim 2023

Araştırma Makalesi/Research Article

*
İnşaat Mühendisliği Bölümü
Manisa Celal Bayar Üniversitesi,
Manisa / Türkiye

Department of Civil Engineering
Manisa Celal Bayar University,
Manisa / Turkey

ORCID: 0000-0002-8895-7602

tugrul.erdem@cbu.edu.tr

**
İnşaat Mühendisliği Bölümü
Manisa Celal Bayar Üniversitesi,
Manisa / Türkiye

Department of Civil Engineering
Manisa Celal Bayar University,
Manisa / Turkey

ORCID: 0000-0001-9971-8546

engin.gucuyen@cbu.edu.tr

ABSTRACT

Superstructure loads are transferred to the foundation by the reinforced concrete (RC) columns, whose design involves sufficient section sizes with longitudinal and transverse reinforcements. However, RC columns are required to be strengthened when low concrete quality and a lack of reinforcement are stated. The most common strengthening technique for the RC columns is jacketing. In this method, the strength of the existing column is improved by concreting and installing jacketing reinforcements. Although columns are designed according to several load combinations, impact loads whose effects may reach very high values at short notice are not regarded. In this study, the impact behavior of the existing and strengthened RC columns is numerically investigated. For this purpose, non-linear incremental dynamic analysis is carried out by Abaqus software, which is widely utilized by engineers for such solutions. Acceleration, displacement, and impact load values, as well as crack patterns, are obtained after numerical analysis. Analysis outputs reveal that proposed finite element models give an opinion about the impact responses of the RC columns. Consequently, it is evaluated that this study may make a contribution to the researchers generating non-linear analysis without having to perform impact experiments that require high costs and heavy workloads in laboratory conditions.

Keywords: Abaqus, Column, Impact behavior, Non-linear analysis, Strengthening

ÖZET

Üst yapı yükleri, boyuna ve enine donatılarla birlikte yeterli kesit boyutlarına sahip olarak tasarlanan kolonlar tarafından temele aktarılır. Ancak beton kalitesinin düşük olduğu ve donatı eksikliğinin belirtildiği durumlarda betonarme kolonlar güçlendirilmesi gerekmektedir. Betonarme kolonlar için en yaygın güçlendirme tekniği mantolamadır. Bu yöntemde, mevcut kolonun dayanımı, beton dökülerek ve mantolama donatıları kullanılarak artırılmaktadır. Kolonlar çeşitli yük kombinasyonlarına göre tasarlanmasına rağmen, etkileri kısa sürede çok yüksek değerlere ulaşabilen çarpma yükleri dikkate alınmamaktadır. Bu çalışmada, mevcut ve güçlendirilmiş betonarme kolonların çarpma davranışları nümerik olarak incelenmiştir. Bu amaçla, mühendisler tarafından bu tür çözümler için yaygın olarak kullanılan Abaqus yazılımı ile doğrusal olmayan artımlı dinamik analizler gerçekleştirilmiştir. İvme, yer değiştirme ve çarpma yükü değerleri ile çatlak dağılımları nümerik analizden sonra elde edilmiştir. Analiz çıktıları, önerilen sonlu eleman modellerinin betonarme kolonların çarpma tepkileri hakkında bir fikir verdiğini ortaya koymaktadır. Sonuç olarak, bu çalışmanın araştırmacıların laboratuvar koşullarında yüksek maliyet ve ağır iş yükü gerektiren çarpma deneylerini yapmak zorunda kalmadan doğrusal olmayan analizler yapmalarına katkı sağlayabileceği değerlendirilmektedir.

Anahtar Kelimeler: Abaqus, Kolon, Çarpma davranışı, Doğrusal olmayan analiz, Güçlendirme

1. INTRODUCTION

Concrete is a structural material comprised of cement, water, and fine and coarse aggregates. Although concrete has several advantages in the construction field, its tensile strength is very low. For this reason, steel reinforcements are used in the tensile areas of the concrete members. Finally, RC structural elements, in which concrete and steel act together to resist internal forces, are utilized in the structures.

RC columns that can be square, rectangular or circle in shape are the vertical carriers in the structural system. Columns are designed to resist vertical and horizontal loads, axial forces, bending moments, and shear forces. Basically, columns receive the loads from the beams and slabs and transfer them to the foundations. Seismic codes pay attention to the design of RC columns by defining the minimum section sizes and reinforcement ratios.

Some buildings need to be strengthened after comprehensively investigating their seismic performances. The most common strengthening method is the jacketing of the existing columns to improve the strength by providing a jacket of additional concrete and steel bars around the existing column (Erdem & Karal, 2022; Sayed et al., 2020). In this method, the plaster and concrete cover of the existing column are removed until the steel bars are observed. Proper grout mortar is used to provide adequate anchoring between concretes and the connection between the steel bars. Jacketing concrete is poured while taking into account the segregation risk. Because of this, surfaces of the existing concrete are made wet in the first place. In addition, the strength of jacketing concrete is supposed to be higher than existing concrete. So, the section of the existing column has become bigger in the end.

Impact loading is one of the least known load effects on structural members. Although impact limits of test devices are published in ASTM E-23 for experimental studies, the design of RC structures exposed to impact loads is not mentioned in the seismic codes (E23-00, 2002). Therefore, researchers have developed test setups with the necessary measurement devices to conduct experimental studies (Anil et al., 2018; Erdem, R. T., 2014). So, the behavior and failure modes of the test specimens produced by different materials have been investigated (Li et al., 2022; Wang et al., 2011; Zhang et al., 2016).

Numerical studies about sudden dynamic load effects have also been performed together with the development in computer technology (Othman & Marzouk, 2017; Genikomsou & Polak, 2015; Li et al., 2017). Thus, the behavior of several structural members resisting impact loads has been a popular study field in recent years (Anil, et al., 2016; Do et al., 2018; Jiang et al., 2012; Zhao et al., 2018). In these studies, impact scenarios which are linked with natural hazards and accidental issues such as sudden explosions, vehicle collisions, rock falls on buildings located in mountainsides, earthquakes, strong winds, landslides, falling of heavy objects on the slabs in industrial buildings, etc. are simulated (Anil, et al., 2018; Yılmaz et al., 2020; Erdem & Gücüyen, 2017).

In this study, the impact responses of the existing and strengthened columns are numerically investigated. For this purpose, the well-known finite elements analysis software called Abaqus is utilized for the non-linear solutions (Abaqus, 2015). Acceleration values around the impact point as well as maximum impact loads and displacements are obtained after analyses. The results are visually presented by time-dependent graphs. Besides, the fracture distributions of the columns are acquired from the software. Researchers usually abstain from incremental dynamic solutions due to the difficulties in the analysis phases, such as material models, mesh sizes, and time increments. So, based on the results of this study, it is evaluated that the analysis outcomes will shed light on the researchers studying the impact behavior of various structural elements.

2. SECTION PROPERTIES AND MATERIALS

Two column sections with different material characteristics and longitudinal reinforcement configurations are taken into consideration. Section sizes of the existing column are 300x300 mm. The values are 50% higher in each direction after strengthening. So, the section sizes of the strengthened column are enlarged to 450x450 mm. The concrete grades of the existing and strengthened columns are C10 and C30 whose compressive strength values are 10 MPa and 30 MPa, respectively.

Column length is considered to be 3000 mm in the analyses. The distance between stirrups (sc) is 100 mm in the bottom and top parts of the column. On the other hand, the stirrup distance (s) is 200 mm in the middle part of the column. The diameters of the longitudinal and transverse steel bars are 14 mm and 8 mm for each column. While there are 8 bars in the existing column, a total of 16 bars are used after strengthening. Section details for the columns are presented in Figure 1. Additional steel bars are marked by the red color in the figure.

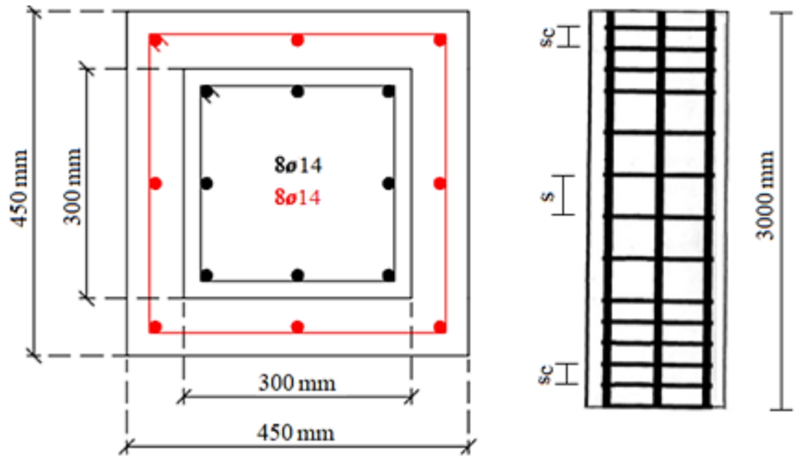

Figure 1. Column details (Self-archive)

3. FINITE ELEMENTS ANALYSIS

The explicit module of the Abaqus software is utilized for numerical simulations. In this module, complex finite element models can be analyzed under the effects of high-speed collisions, contact, and large deformations. Besides, various material models can be adapted to the numerical models. First of all, the geometries of the finite element models are created by using the most convenient element types under impact loading in the software. Afterwards, material properties are assigned to the related sections. Important analysis parameters, such as finite element size and time increments, are also defined. Finally, the analysis is performed by considering the contact properties between surfaces. The main steps for the incremental dynamic analysis are shown in Figure 2.

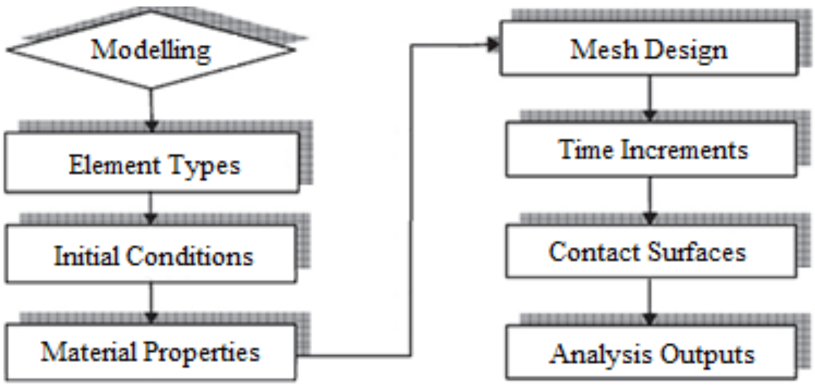

Figure 2. Analysis steps (Self-archive)

A 10-node modified tetrahedron-shaped element (C3D10M), which is widely used for impact simulations, is used in the library of Abaqus, as given in Figure 3 (Abaqus, 2015). Thus, three-dimensional models are created. Drop height, mass of the striker that implements impact loads on the columns, reinforcement configuration, as well as support conditions are defined in the software. Edge parts of the columns are considered to be fully fixed supports at the top and bottom surfaces.

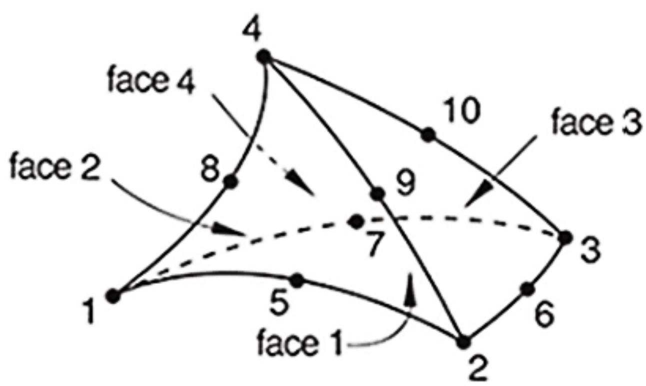

Figure 3. Element properties (Abaqus, 2015)

As the existing and strengthened columns are considered, two different concrete grades are taken into consideration. For this purpose, a concrete damage plasticity model that represents the behavior of concrete in the compression and tension regions is used. In this model, nonlinearity of concrete is defined due to the stress-strain relationship as presented in Figure 4.

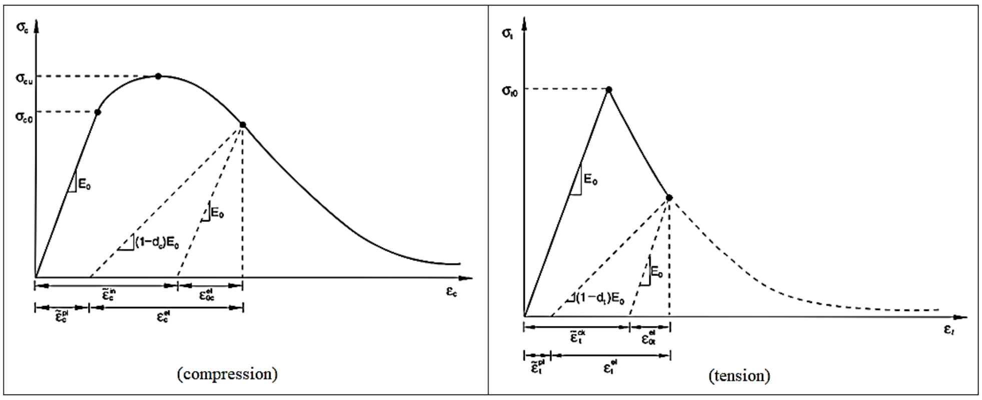

Figure 4. Material model (Abaqus, 2015)

The concrete damage plasticity model is a continuum, plasticity-based damage model offered by Abaqus that follows the stress-strain relationship of concrete by considering the uniaxial tensile and compressive response. In the compression part, the path behaves linearly to the value of the initial yield (σc0). After that, the plastic behavior is modeled owing to the stress-hardening and the strain-softening starting beyond the ultimate stress (σcu). In the tension region, the stress-strain relationship is linear until the failure stress value (σt0), where the first micro-crack occurs. Beyond this value, a softening stress-strain path is observed.

Elastic modulus, compressive and tensile strengths, Poisson's ratio, and density values of concrete, as well as other plasticity values such as dilation angle (ψ), the eccentricity of the flow potential (e), the ratio of initial equibiaxial compressive yield stress to initial uniaxial compressive yield stress (σb0/σc0), the coefficient of the deviatoric cross-section shape (Kc), and the viscosity parameter (μ), are defined in the software. In addition, the final strain value of concrete (εcu) is defined as 0.003 according to Mander's material model. The values for both concrete grades are given in Table 1.

Table 1. Material properties of concrete

Property	Value for C10	Value for C30
Poisson's Ratio	0.20	0.20
Density (kg/m^3)	2400	2400
Elastic Modulus (MPa)	14862.70	25742.96
Compressive Strength (MPa)	10	30
Tensile Strength (MPa)	1.97	3.41
ψ	30	30
e	0.10	0.10
σ_{b0}/σ_{c0}	1.16	1.16
K_c	0.6667	0.6667
μ	0.0001	0.0001

The linear elastic material model is used to define the steel striker. On the other hand, an elastic-perfectly plastic material model is utilized for steel reinforcement characteristics in the software. To model the adherence between the columns and steel reinforcement, the tie contact property is employed in the software. Material characteristics are introduced in Table 2.

Table 2. Material properties of steel

Property	Reinforcement	Hammer
Poisson's Ratio	0.30	0.30
Density (kg/m^3)	7850	7850
Elastic Modulus (MPa)	200000	200000
Shear Modulus (MPa)	76923	76923
Bulk Modulus (MPa)	166670	166670
Yield Strength (MPa)	420	-

Finite element models are distributed into smaller pieces to obtain more accurate analysis outputs. For this purpose, a sensitivity analysis is performed between 10- and 20-mm element sizes to determine the finite element size by considering the computer limits and computational time. Due to the analysis results, there are no significant differences around the value of 15 mm. So, finite element size is taken as 15 mm in the analyses. A steel loading plate is placed at the impact point to prevent local crushing. While 37607 number of nodes and 26209 number of elements are used for the existing column, 58689 number of nodes and 41239 number of elements are utilized for the strengthened column. The vertical and horizontal movements of the support regions that are defined at both ends of the columns are restrained in the software. Numerical models of the columns before and after mesh design are presented in Figure 5.

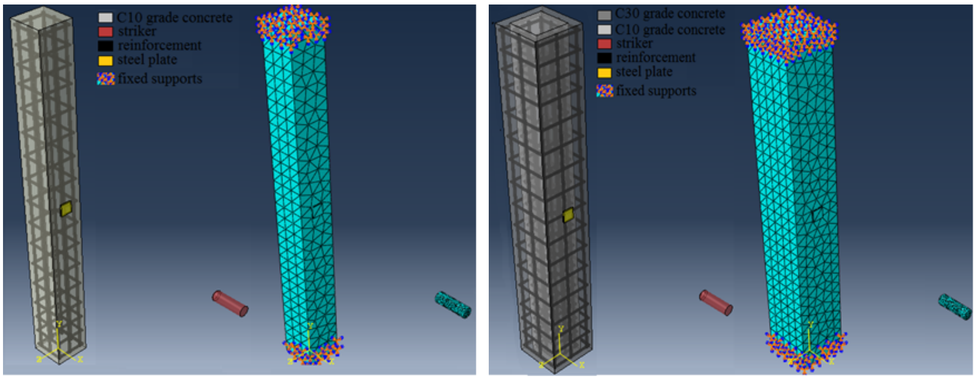

Figure 5. Numerical models of the columns (Abaqus, 2015)

Time steps and increments are the other remarkable parameters in the incremental dynamic analysis. Time steps involve the entire motion of the striker until applying the impact loading to the columns. On the other side, time increments become more important when contact occurs between the striker and the column. The value of time increments is utilized as 2×10^{-8} sec from the beginning of the contact moment.

The connection between the striker and the columns must be defined properly by selecting the related surfaces of the finite element models through the surface-to-surface contact property of the software. In addition, the surfaces of the striker and the columns are defined as master and slave, respectively. To simulate the probable friction losses during the impact motion, the friction coefficient is considered to be 0.02 for the tangential behavior of the contact surfaces.

To apply the same input energy to the columns, the drop height and mass of the striker are taken as constants in the analysis. Low-velocity impact behavior is simulated for both cases of the columns. For this purpose, the values of the drop height and mass of the striker are 2500 mm and 50 kg in the software. Thus, the total impact energy on the columns (mgh = 50x9.81x2.5) is calculated as 1.226 kJ.

In the experimental studies, accelerometers are placed at a specific distance from the impact point to prevent possible damages in the measurement devices. In addition, dynamic load cell is situated in the edge part of the striker to obtain the maximum impact load. Displacement transducers are usually fixed in the lower surfaces of the specimens around the impact point to obtain the residual displacement values on the specimen. In this study, a similar procedure in the experimental studies is followed. Thus, acceleration values are determined from a distance of 100 mm from the impact point. Also, maximum displacements and impact load values are obtained after performing non-linear numerical analyses. The results are given in Table 3 for the existing and strengthened columns.

Table 3. Numerical results

Column Type	Acceleration (g)		Displacement (mm)	Impact Load (kN)
Existing Column	-427	+451	19.52	41.73
Strengthened Column	-496	+534	8.39	43.08

Acceleration-time, displacement-time, and impact load-time graphs are also generated for each situation to visually present the effect of the strengthening technique on the existing column. The graphs are presented in the figures below.

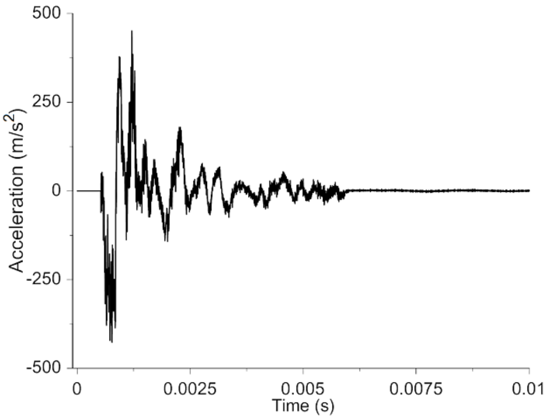

Figure 6. Acceleration-time graph for the existing column (Self-archive)

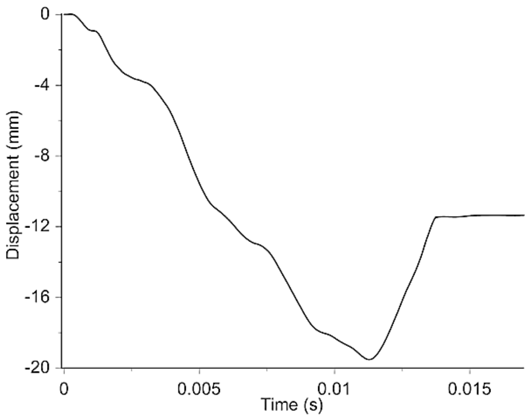

Figure 7. Displacement-time graph for the existing column (Self-archive)

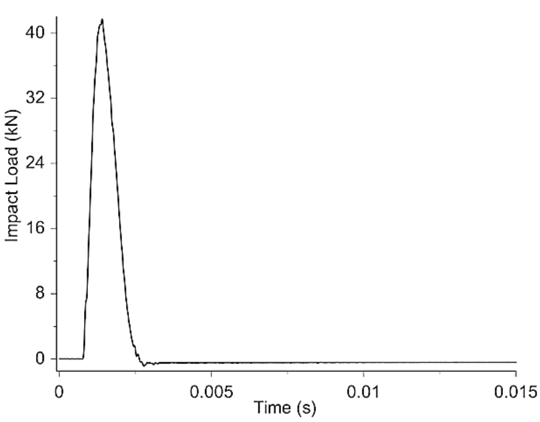

Figure 8. Impact load-time graph for the existing column (Self-archive)

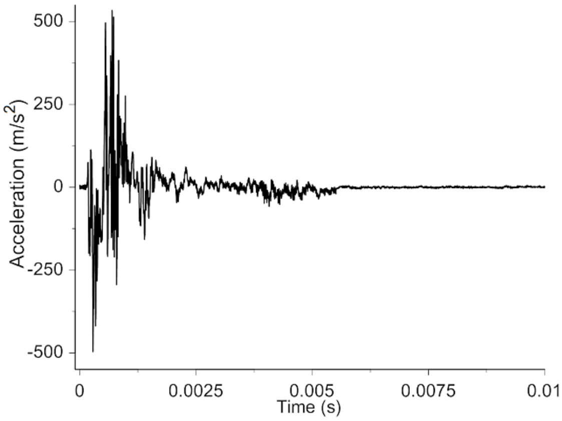

Figure 9. Acceleration-time graph for the strengthened column (Self-archive)

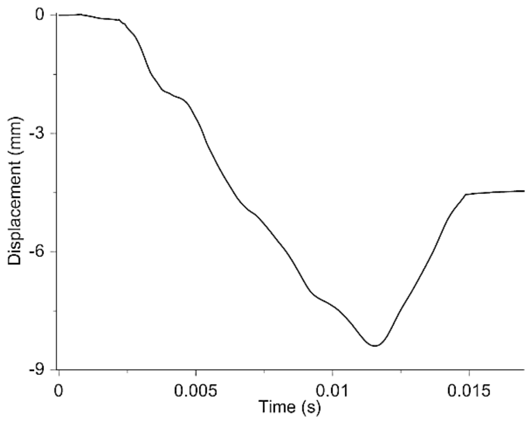

Figure 10. Displacement-time graph for the strengthened column (Self-archive)

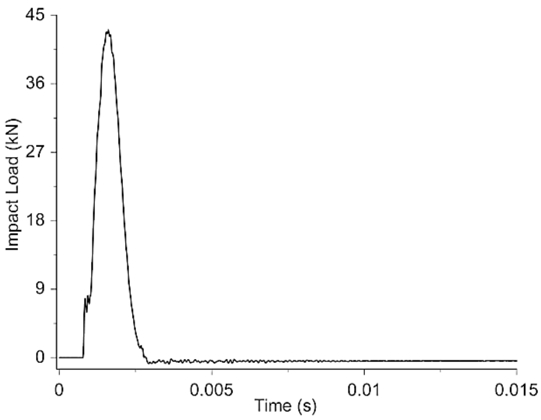

Figure 11. Impact load-time graph for the strengthened column (Self-archive)

In the final step of the numerical analysis, crack formations in the columns are acquired from the software. In line with this purpose, the damaget property of the software is used. This property yields the damage patterns under impact loading. Figure 12 shows that the maximum damage is accumulated around the impact point for each situation. However, a larger area is affected for the existing column.

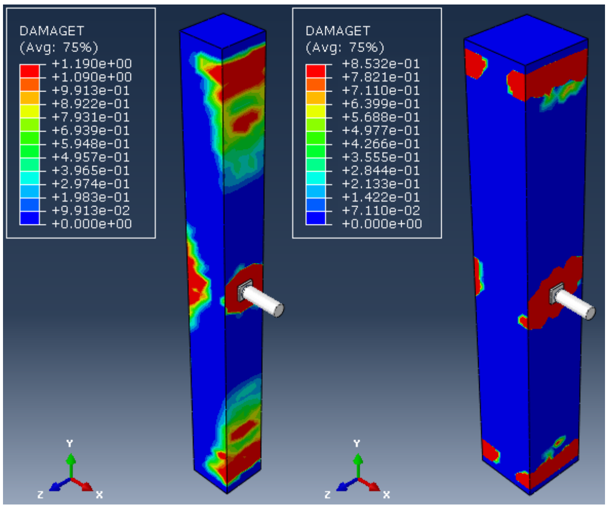

Figure 12. Damage patterns (Abaqus, 2015)

4. CONCLUSOINS

Past major-scale earthquakes have revealed that the strength of RC columns has a significant effect on the safety of the structural system. Therefore, columns are designed according to various load combinations in the seismic codes. However, the effect of impact loading on the columns is ignored in the design of RC structures. The jacketing technique comes forward as a rapid and effective strengthening method in the low-capacity columns. Basically, this method involves the addition of new steel bars and high-strength concrete. The subject of this study becomes more interesting in terms of the applied load type on the existing and strengthened columns.

Since there is a lack of detailed studies about the impact effects of structural members, this study aimed to present the impact behavior of existing and strengthened RC columns. As generating test setups is time-consuming, providing perfect support conditions is not easy, and the costs of test devices are high, numerical solutions appear as alternative methods to investigate the impact behavior of structural members. For this purpose, non-linear incremental analyses are performed under constant impact energy by the well-known software Abaqus. Analysis outputs such as acceleration, displacement, and impact load values are obtained and visually exhibited by time-dependent graphs for each column to track the dynamic responses.

When the analysis results are investigated, it is seen that the strengthening method is effective in limiting the maximum displacement of the existing column. As the strength of the existing column is improved after jacketing, bigger acceleration values are obtained for the strengthened column. On the other hand, there aren't big differences between the maximum impact loads resulting from the fixed drop height and mass of the striker values.

Finally, damaget property of Abaqus has been used to determine the possible cracks and failure modes of the RC columns under impact loading. Thus, damage patterns of the columns can be followed after numerical analysis. It is considered that the rigidity of the columns is an important parameter in damage distributions. Although maximum damages are observed around the impact point, a larger area of the existing column is affected under impact loading, as expected.

As the manufacture of RC test specimens requires attentive workmanship, experimental studies necessitate the proper design of impact test setups, and test devices have measurement limits, numerical simulations reduce workloads when accurate analysis steps are provided with the help of computer technology. Thus, this study is considered useful for the engineers studying the impact behavior tendency of different structural members. Applying different strengthening options in terms of shape, reinforcement, and concrete could be utilized in future studies.

REFERENCES

Abaqus Version 6.12 (2015). User's Manual. Simulia, Dassault Systèmes Simulia Corp.

E23-00, A. (2002). Standard test methods for notched bar impact testing of metallic materials. ASTM International.

Erdem, R. T. (2014). Prediction of acceleration and impact force values of a reinforced concrete slab. Computers and Concrete, 14(5), 563–575.

Wang, X., Zhang, Y., Su, Y. & Feng, Y. (2011). Experimental investigation on the effect of reinforcement ratio to capacity of RC column to resist lateral impact loading. Systems Engineering Procedia, 1, 35–41.

Anil, Ö., Erdem, R. T. & Tokgöz, M. (2018). Investigation of lateral impact behavior of RC columns. Computers and Concrete. 22(1), 123–132.

Li, H., Chen,W., Huang, Z., Hao,H., Ngo, T. T. & Pham, T. M. (2022). Influence of various impact scenarios on the dynamic performance of concrete beam-column joints. International Journal of Impact Engineering, 167, 104284.

Zhang, X., Hao, H. & Li, C. (2016). Experimental investigation of the response of precast segmental columns subjected to impact loading. International Journal of Impact Engineering, 95, 105–124.

Do, T. V., Pham, T. M. & Hao, H. (2018). Numerical investigation of the behavior of precast concrete segmental columns subjected to vehicle collision. Engineering Structures, 156, 375–393.

Jiang, H., Wang, X. & He, S. (2012). Numerical simulation of impact tests on reinforced concrete beams. Materials & Design, 39, 111–120.

Zhao, D. B., Yi, W. J. & Kunnath, S. K. (2018). Numerical simulation and shear resistance of reinforced concrete beams under impact. Engineering Structures, 166, 387–401.

Othman, H. & Marzouk, H. (2017). Finite-element analysis of reinforced concrete plates subjected to repeated impact loads. Journal of Structural Engineering, 143(9), 1–16.

Genikomsou, A. S. & Polak, M. A. (2015). Finite element analysis of punching shear of concrete slabs using damaged plasticity model in abaqus. Engineering Structures, 98, 38–48.

Li, J., Hao, H. & Wu, C. (2017). Numerical study of precast segmental column under blast loads. Engineering Structures, 134, 127–137.

Anil, Ö., Erdem, R. T. & Tokgöz, M. N. (2018). Investigation of lateral impact behavior of rc columns. Computers and Concrete, 22(1), 123–132.

Erdem, R. T. & Gücüyen, E. (2017). Non-linear analysis of reinforced concrete slabs under impact effect. Gradevinar, 69(6), 479–487.

Yılmaz, T., Kıraç, N., Anil, Ö., Erdem, R. T. & Kaçaran, G. (2020). Experimental investigation of impact behaviour of rc slab with different reinforcement ratios. KSCE Journal of Civil Engineering, 24(1), 241–254.

Anil, Ö., Durucan, C., Erdem, R. T. & Yorgancular, M. A. (2016). Experimental and numerical investigation of reinforced concrete beams with variable material properties under impact loading. Construction and Building Materials, 125, 94–104.

Erdem, R. T. & Karal, K. (2022). Performance evaluation and strengthening of reinforced concrete buildings. Revista de La Construcción, 21(1), 53–68.

Sayed, A. M., Rashwan, M. M. & Helmy, E. H. (2020). Experimental behavior of cracked reinforced concrete columns strengthened with reinforced concrete jacketing. Materials, 13(12), 2832.