IRAQI Academic Scientific Journals

SAMARRA JOURNAL OF ENGINEERING SCIENCE AND RESEARCH

S.J.E.S.R

Assessing Shear Behaviour in Reinforced Concrete Beams: A Numerical Approach Using a Finite Element Program (ABAQUS)

Othman Majeed Abdullah[1*], Aziz Ibrhim Abdulla[2] and Wisam Amer Alus[3]

1- Department of Civil Engineering, College of engineering, University of Tikrit, Iraq
2- Department of Civil Engineering, College of engineering, University of Tikrit, Iraq
3- Department of Civil Engineering, College of engineering, University of Tikrit, Iraq

Article Information

Received: 27/04/2024
Accepted: 03/06/2024

Keywords:
ABAQUS, Concrete Beam, Cracks, Finite Element, Mode Failure.

Corresponding Author

E-mail:
engothman25@gmail.com

Abstract

Structural engineers frequently employ numerical techniques to provide approximate solutions to complex problems. This methodology is known as the finite element approach. It often collapses structural components into very small parts. Fundamental concepts of structural analysis and design theory are successfully included in nonlinear finite element analysis, which frequently yields accurate predictions of structural behavior. Recent advances in computer technology have contributed to the rise in popularity of finite element analysis. This article discusses the creation of the 3D model of nonlinear finite element for RC beams with shear behaviour. We generated and evaluated the finite element models using the ABAQUS program, specifically version 2019. The findings of the Finite Element Analysis for the strain distribution, load deflection relationship, modes of failure, and load capacity were mostly consistent with the experimental data. The experiment's actual maximum load capacity was, on average, 2.75%, lower than the figure that the ABAQUS computer algorithm projected. The practical trial and computational results showed a mean deflection difference of 7.54% at ultimate loads. Finite element (FE) analysis might be a good way to make estimates based on the beams' cracking behaviour, load-bearing capacity, and deformation characteristics. This is due to its ability to properly simulate crack propagation and structural damage, resembling the outcomes of actual investigations.

1. Introduction

The Finite Element (FE) approach is employed by the ABAQUS program to investigate the structural response of reinforced concrete (RC) beams under a range of parameter circumstances. A wide variety of element types and material properties are available in the ABAQUS computer system, which may be used to precisely model different geometries and replicate the linear and nonlinear behaviour of different designed materials [1]. One composite material that is frequently utilized in engineering constructions is concrete. Experimental studies have demonstrated that under uniaxial compression, it displays notable nonlinearity. The RC in the tensile strength is equivalent to 1/10 percent of the compressive strength. The flexural cracks start at the side of the material experiencing tension and may extend, at most, until reaching the neutral axis. Cracking in reinforced concrete components is often characterized by its complicated structure. It includes several methods and characteristics related to the interaction between concrete and reinforcement, as well as factors like geometry, type of stress, and support conditions. In this study, the finite element method (FEM) was employed to provide a numerical solution for the failure behaviour of reinforced concrete (RC) structures. ABAQUS, a widely used commercial software, incorporates a plasticity damage model specifically designed for RC, offering a cost-effective alternative to experimental approaches. The FEM was utilized to analyse the non-linear behaviour of materials in reinforced concrete structures. Cracking represents a critical concern in the structural performance of buildings, as it can manifest in various elements, including walls, beams, columns, and slabs, primarily due to strains that are often neglected during the design phase.

The material properties of the structural elements of reinforced concrete beams, conditions of support, methods of load application, and interactions on the surface of the beam are the subjects of this work. The load-deflection curve of the models was tabulated and compared with the experimental result in determining the accuracy of the reinforced concrete beam models in terms of cracking pattern performance.

2. Literature Review

Several studies have used finite element software (such as ABAQUS [2], ANSYS, and LS-DYNA) to look at how simply supported reinforced concrete beams react to monotonic loading at different rates.

Ali Ahmed [3] conducted an examination of the impact of the dynamic load test on the beam element of a structure, drawing on Kishi's research. He estimated the dynamic responses under pressure using a 3D finite element (EF) simulation in ABAQUS. The research's findings and observations have been focused on tension damage (cracks) with superimposed experimental beam cracks and concrete stiffness degradation contour plots of the half of the beam where the cracks value was between 0 and 0.4 mm, indicating that if the cracks values varied from 0.4 mm to 1mm, the concrete would fail in laboratory experiments.

In this study, researchers compare the experimental and ABAQUS results for reinforced concrete beam bending failure analysis reported by [4]. According to the findings of the investigation, the beam in the elastic stage before being loaded with .83 % from ultimate load it has high stiffness and strength. Furthermore, load levels increase linearly with deflection as the plastic stage begins. After the load achieved 24 kN, the deflection was 10.521 mm Abaqus and 12.795 mm experimental. The study at [5] focused on the analysis of RCC beam members using ABAQUS. Researchers have performed a comparison of numerical and experimental data. This research applied different reinforcement ratios and discovered that over-reinforced beams can support more load than other beams, with the firstn crack occurring at 36 kN. Due to higher tension stress, under-reinforced beams outperform balanced and over-reinforced beams. Shama Al Hasani et al. [6] presented a numerical study of reinforced concrete beams using ABAQUS. The validation of the model's accuracy against experimental data revealed a close agreement, particularly in terms of displacement at ultimate and failure loads. The maximum displacement for the present study was 11 mm, while the experimental result was 12.795 mm, indicating a comprehensive agreement rate of approximately 86%. The results provided valuable insights into the behavior of RC beams under load. Recommendations for using a fine mesh and smaller load increments were made to achieve more accurate results. Jadhav et al. [7] investigated the mechanism of concrete cracking and propagation owing to reinforcement corrosion in RCC, and their theoretical findings were compared to Hankare's experimental results [8]. They tested three different corrosion percentages:

1.5%, 4.5%, and 12%. The results demonstrate a satisfactory agreement with the experiments. The corrosion will reduce the load capacity by 1.5%. However, 4.5% or more will reduce capacity. Mundeli Salathiel et al. used finite element modeling to study the repair and strengthen reinforced concrete beam patches using fiber-reinforced polymers. '[9]' This study's primary objective was to use ABAQUS software to analyze the behavior of reinforced concrete beam patches that had been strengthened and repaired using fiber-reinforced plastic (FRP) composites and to compare the findings to those of experiments. This study indicates that when the maximum Principal stress is greater than the tensile strength of the concrete, cracking happens. In terms of load-deflection relationships, the finite element analysis findings were quite close to the experimental results. Gilberto Rodríguez et al. [10] conducted a study on reinforced concrete beams enhanced with FRP bars and GFRP sheets under impact loading conditions. Nine beams were fabricated and evaluated under various configurations and loading situations utilizing ABAQUS software and experimental work. The experimental results and the finite element model results using ABAQUS are quite similar. The greatest displacements in ABAQUS were 85.43 mm, which represented only a 3.8% deviation from the experimental output of 82.3 mm. Huang, et al [11] investigated strengthening reinforced concrete beams with FRP bars under impact loading, which successfully improves their capacity performance and reduces the progression of damage. Wani and Mohammod [12] discovered that the tensilen strength of theo concrete increased by an average of 7.5% when 5% of the cement was replaced with waste paper pulp, and then dropped gradually after 10% replacement. Jyotirmoy and Panigrahi's [13] investigation of fiber-reinforced geopolymer concrete (FRGC) is examined, and it is indicated that FRGC has high thermal stability, is lightweight, and has low shrinkage. Thus, rapid innovation in fiber-reinforced geopolymers is expected shortly. Deng Sihua et al. [14] investigated the nonlinear analysis of reinforced concrete beam bending failure experimentation Based on ABAQUS, the researchers effectively developed and verified an ABAQUS numerical model for predicting crack propagation in RC beams. The model accurately simulated the nonlinear behavior of RC beams under bending stresses. The findings serve as a useful reference for future studies on RC beam analysis, advocating fine mesh and lower load increments for increased accuracy. The theoretical (numerical) and experimental findings had a thorough agreement rate of roughly 86%, confirming the suggested model's good correlation and dependability.

2. INPUT DATA

The constitutive models for concrete damaged plasticity (CDP) and concrete smeared cracking (CSC) are both included in Abaqus [15]. Because of its increased stability, the CDP model is recommended over the CSC model. The CDP model, which is frequently used to characterize the mechanical response of reinforced concrete (RC) under various loading circumstances, is incorporated into ABAQUS [6]. In the present study, the damage plasticity model was applied. The material characteristics shown in Table 1.

Table 1. Material characteristics of the tested beams

Where:

B: Beam Symbol (1, 2, 3: Group Number)

S: Shear failure

C: Combined failure

F: Flexural failure

Symbol	a/d	Shear reinforcement spacing (mm)	Pu (KN)	Mode Failure
B1S	2.44	75	189.6	Shear Failure
B2S		100	186.034	Shear Failure
B3S		150	174.76	Shear Failure
B1C	3	75	167.92	Combined Failure
B2C		100	150.14	Combined Failure
B1F	3.571	75	128.16	Flexural Failure
B2F		100	124.97	Flexural Failure
B3C		150	132.91	Combined Failure
B3F	4.571	150	113.106	Flexural Failure
B4S1	2.44	300	142.87	Shear Failure
B4S2	3	300	136.681	Shear Failure
B4S3	3.571	300	109.868	Shear Failure

4. MODELING OF REINFORCED CONCRETE BEAM

4.1. Geometry Modelling

The Cantilever beam presented in this study is an RC beam with cross-section dimensions (200*300) mm and effective depth (266) mm in Figure 1. In ABAQUS, the reinforced material utilized the T3D2 element, and the concrete, the C3D8R element. In order to mimic the bonding interaction between the reinforcement and concrete, we inserted it into the reinforced concrete.

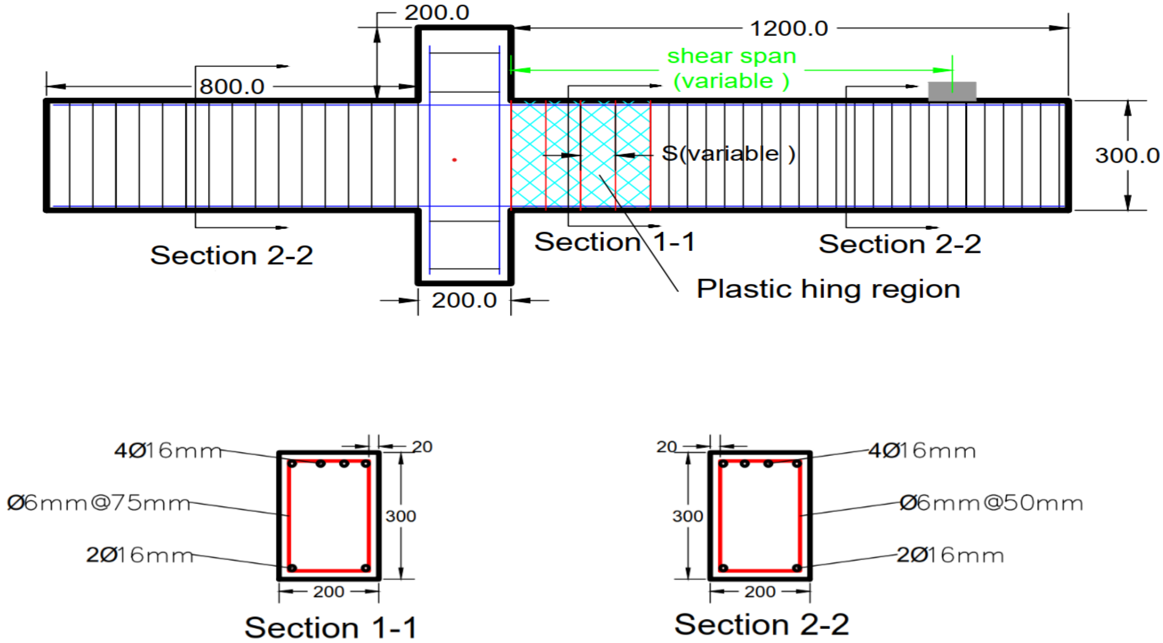

Fig. 1. Modelling of the beam

4.2 Model Meshing

After assembling the module, we evaluated the computer programme's performance and the accuracy of the experiment results to determine the optimal mesh size for the reinforced concrete beam models. Figure 2 depicts the concrete element's diameter at 15 mm, as well as the longitudinal reinforcing bar and transverse reinforcement stirrup mesh's diameter at 20 mm.

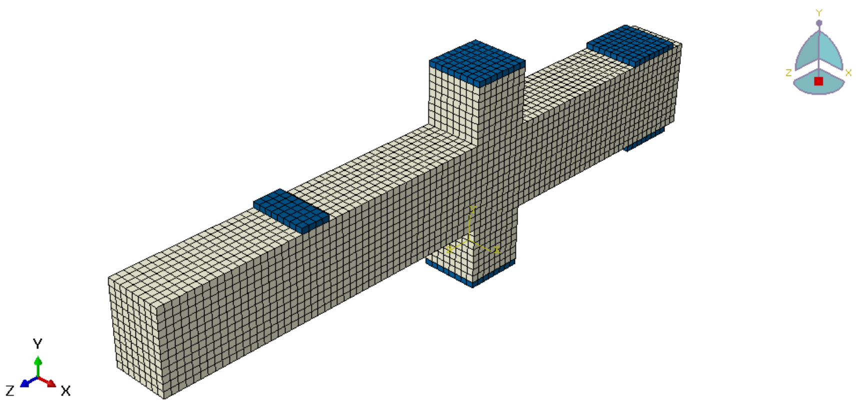

Fig. 2. The meshing of the beam

4.3. SURFACE INTERACTIONS

It is common knowledge that one of the most significant factors that significantly influences the accuracy of results is the modeling of the interaction between many components. With the "embedded region" function in the ABAQUS computer software, the steel reinforcement and stirrups were carefully simulated how they would fit into the surrounding concrete to provide a strong connection. Nonetheless, the "penalty" friction formula with a friction coefficient of 0.6 [16] and the "tie" constraint were used in this study to model the interaction between the concrete and the supports. Because of the use of this limitation, these members stay fully connected and keep up with operations throughout the. analysis.

5. Applying loads and boundary conditions.

A pressure load was delivered by each group to the upper surface of the steel loading plates, which measured 100 by 200 mm. Using the same configuration as the reinforced concrete beam experiment, two metal plates masses were positioned at a distance equal to the shear span from the support to replicate the application of pressure. For every RC beam in this investigation, the displacement boundary conditions were simulated to correspond with the experimental configuration. This required that the beam be subjected to static loads while it was configured as a cantilever beam. limited the movement of the support to one line that ran the whole width of the reinforced concrete beam in all directions. In this case, we set the displacements Ux, Uy, and Uz to zero, effectively treating it as a "fixed" support. Under static load, boundary conditions refer to the specific constraints or limitations imposed on a system or structure when subjected to external loads that do not change over time. Apply a concentrated load to the plate, as shown in ‘Fig. 3’. The plate, which measures (50*100) mm, is located at a predetermined supporting distance. Table 2 presents the characteristics of plasticity that the specimens have influenced.

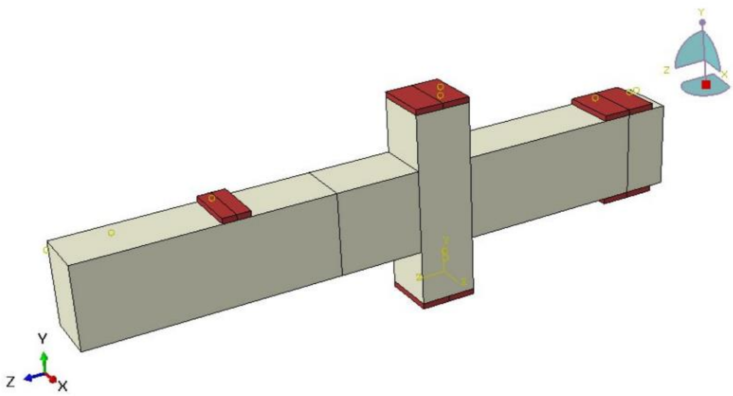

Fig. 3. Loading and constraints

Table 2. Damaged plasticity parameters

Plasticity	Value
Dilation angle	30
Eccentricity	0.1
fb0/fc0	1.16
K	0.6667
Viscosity parameter	0
Density of concert	$\mathbf{2.4} \times \mathbf{10{-}6}$N/mm3
Density of steel	$\mathbf{7.65} \times \mathbf{10{-}5}$N/mm3
Poisson's ratio of concrete	0.2
Poisson's ratio of steel	0.3
Concrete strength	C25

6. RESULTS AND DISCUSSION

6.1. Load Displacement Curve

As seen in Figure 4, the load-displacement curve derived from the current investigation is contrasted with the outcomes of the finite element (FE) analysis. The predictions made by the finite element analysis are remarkably accurate and nearly match the outcomes of the particle tests. On average, the experimental ultimate load capacity was marginally less than the number produced by the ABAQUS software. At maximum loads, the average difference in deflection between the theoretical and actual findings was 7.54%. The reinforced concrete beams' failure modes were well-represented by the constitutive models. The ultimate loads and maximum displacement under failure are shown in Table 3.

Table 3. Summary of the experimental and finite element results for all the tested beams

Symbol	and	Pu (KN)		Expn/FE Ultimate load ratio	Deflection (mm)		Expn/FE Deflection ratio	Failure Modes
		Expn.	FE		Exp.	FE		
B1S	2.44	189.6	194.882	1.027	26.816	30.8168	0.87	
B1C	3	167.92	170.575	0.984	82.94	79.8043	1.039	S. F
B1F	3.571	128.16	136.828	0.937	80.2	65.2439	1.229	C.F
B2S	2.44	186.034	177.018	1.051	27.622	27.416	1.008	F.F
B2C	3	150.14	162.791	0.922	77.61	70.3077	1.104	S. F
B2F	3.571	124.97	132.987	0.940	72.223	72.1228	1.001	C.F
B3S	2.44	174.76	175.389	0.996	29.79	32.5886	0.914	F.F
B3C	3.571	132.91	139.041	0.956	73.93	51.1403	1.446	S. F
B3F	4.571	113.106	110.458	1.024	83.7	78.0637	1.072	C.F
B4S1	2.44	142.87	152.834	0.935	15.952	15.2729	1.044	F.F
B4S2	3	136.681	139.066	0.983	24.64072	24.8848	0.990	S. F

B4S3	3.571	109.868	114.741	0.958	31.0095	31.538	0.983	C.F

S.F: Shear Failure, C.F: Combined Failure: F.F: Flexural Failure

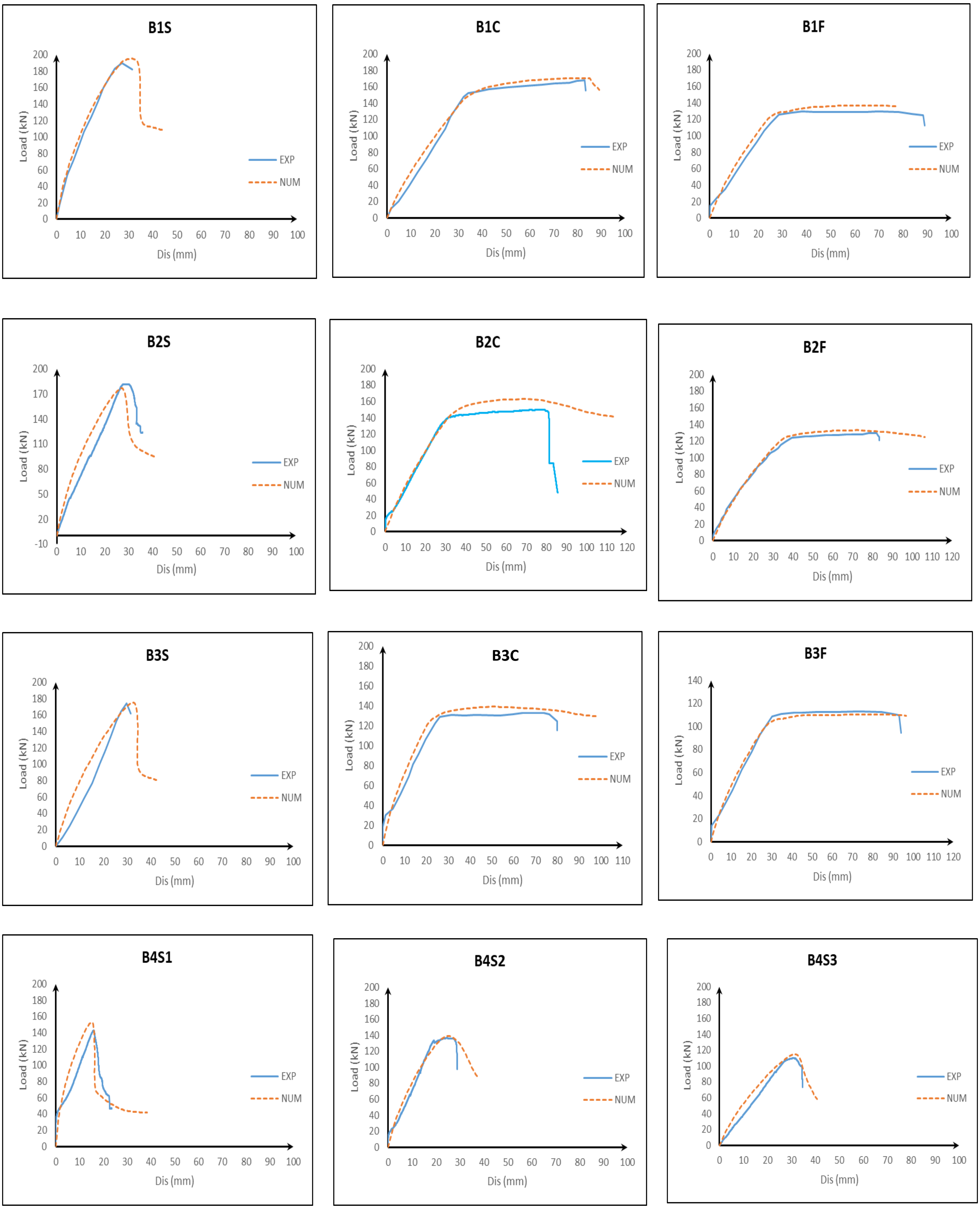

Fig. 4. Load-displacement curve

6.2. Failures Mode of the Examined Beams

A comparison between the failure modes seen in reinforced concrete beams during testing for all groups and those anticipated by the numerical analysis is presented in Figures 5–8. The numerical analyses successfully predicted the experimentally observed concrete crushing. The finite element calculations showed that diagonal cracks happened in all of the specimens. These cracks were caused by tension splitting in the structure that connected the load to the support. These figures illustrate the level of agreement between the numerical analysis and the experimental data across the entire range of a/d ratios.

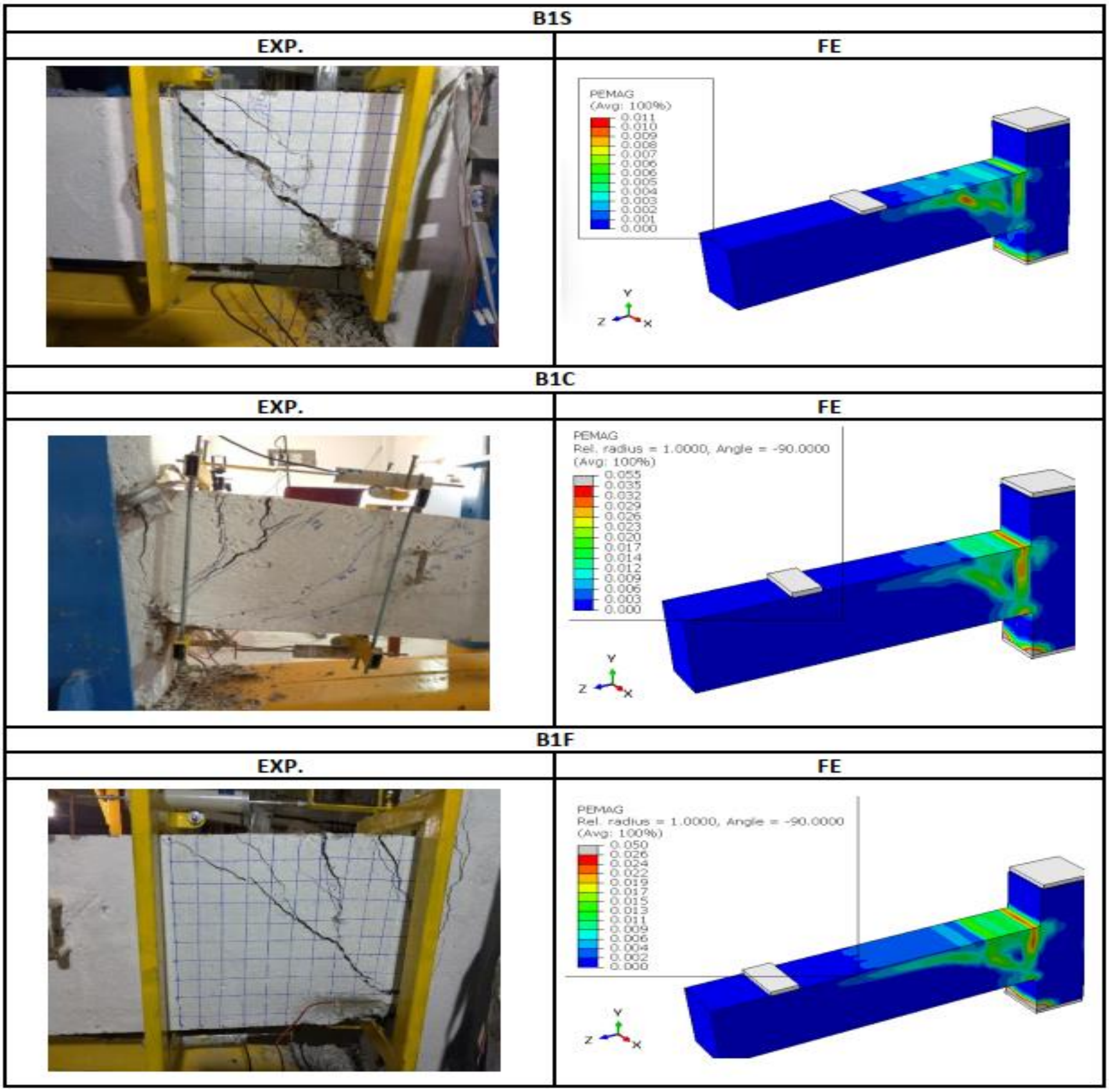

Fig. 5. Compression failure mode for the first group

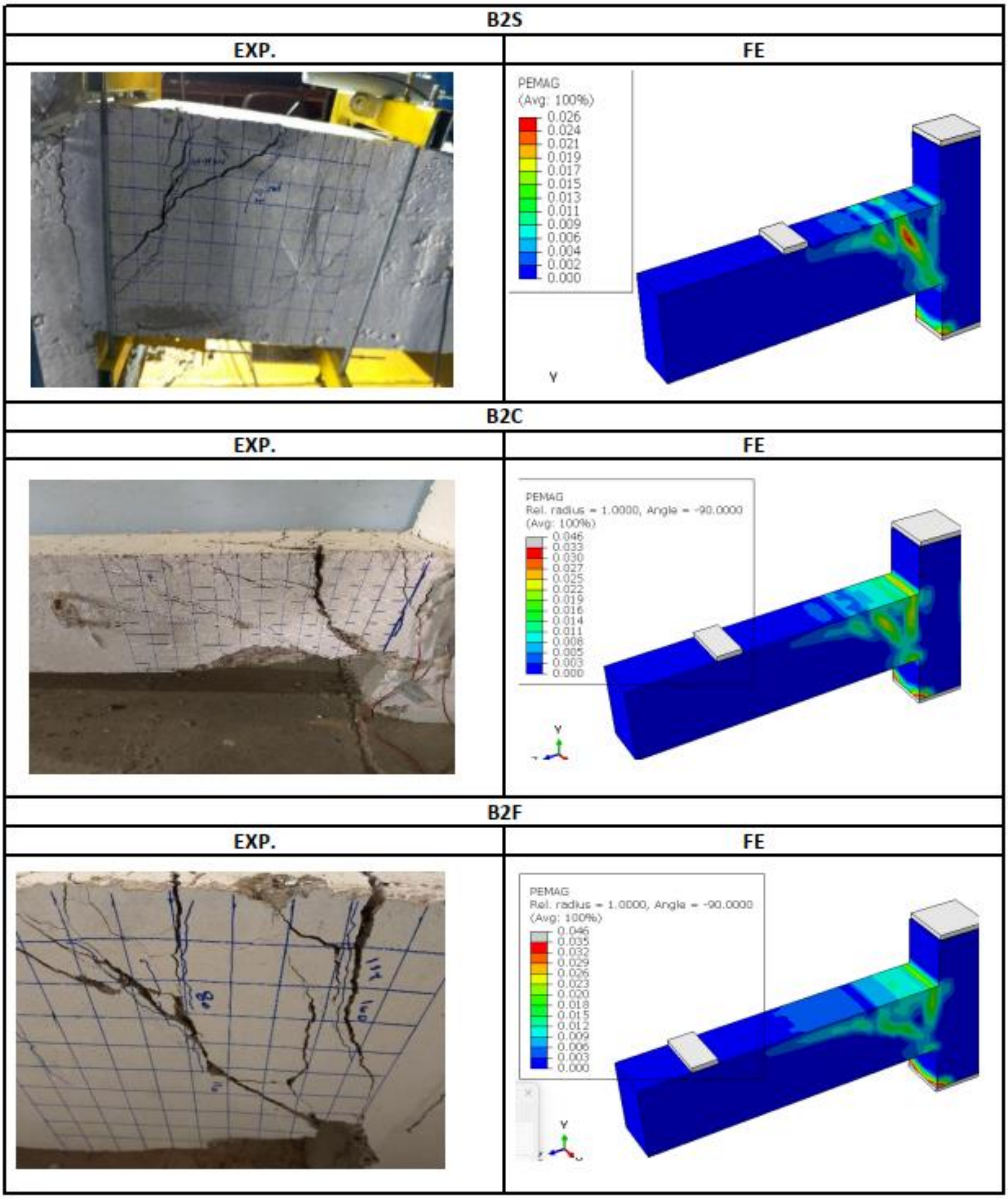

Fig. 6. Compression failure mode for the second group

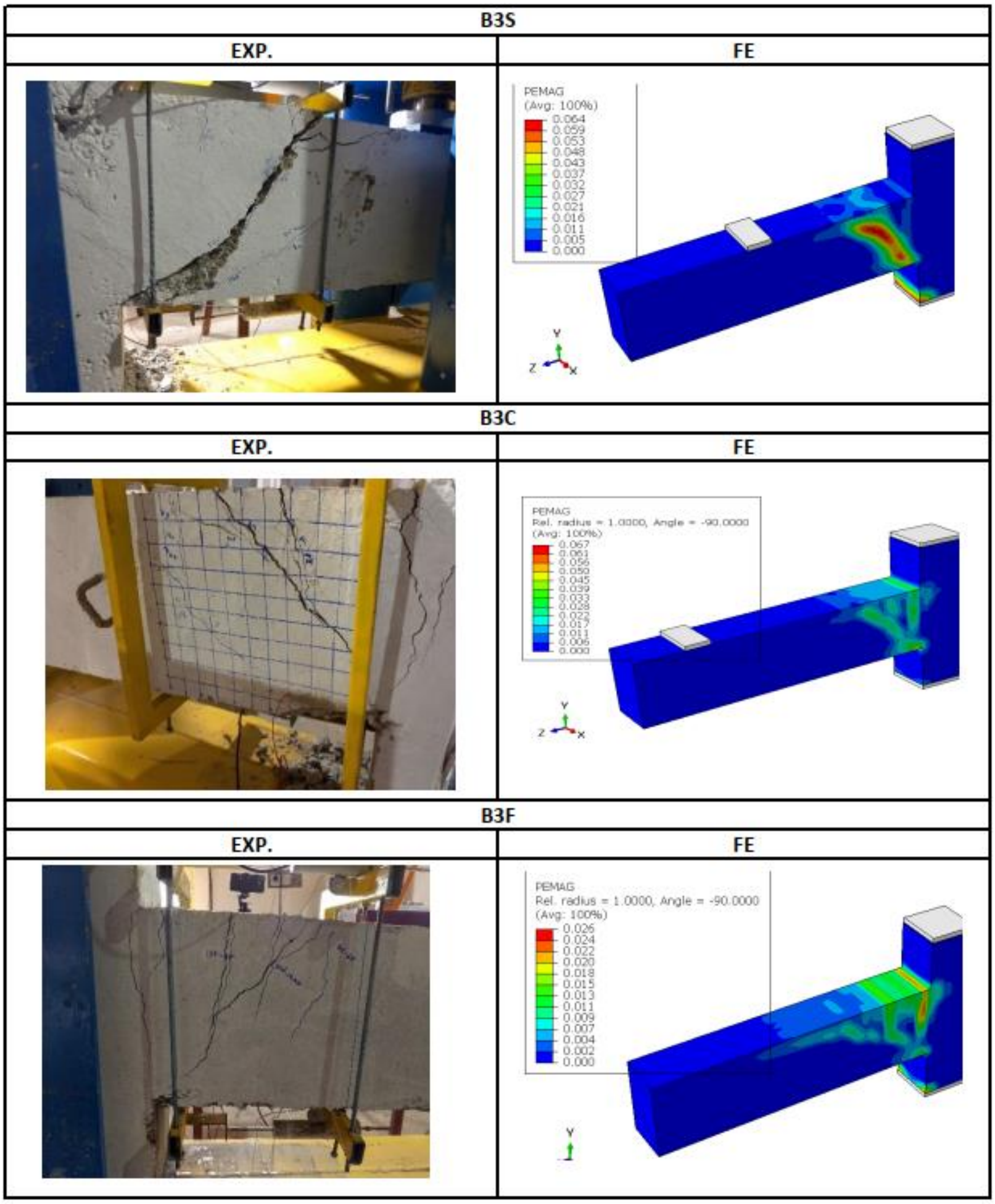

Fig. 7. Compression failure mode for the third group

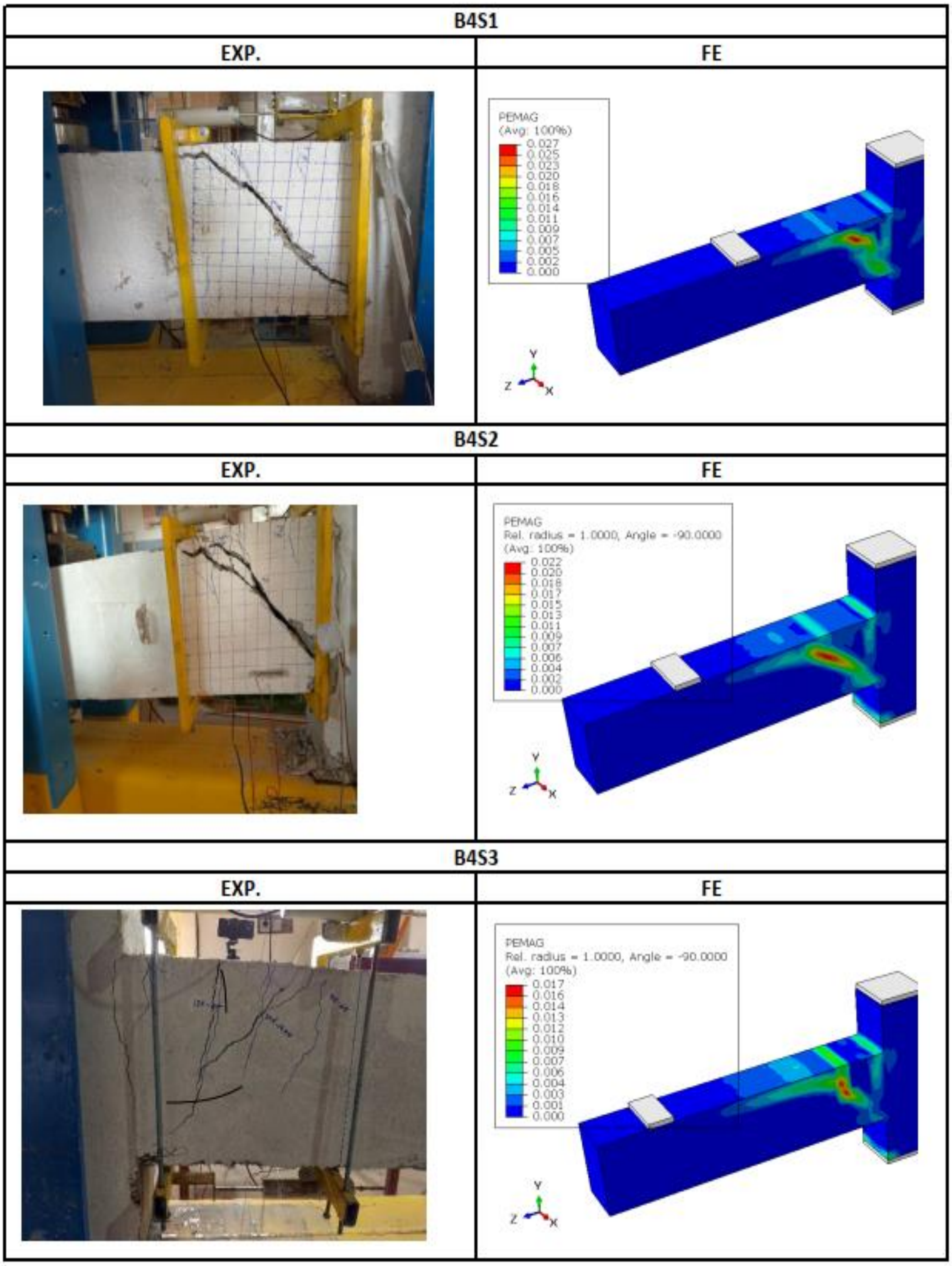

Fig. 8. Compression failure mode for the fourth group

7. Conclusion

The findings of the finite element analysis, with regard to overall behavior, failure mechanism, load deformation, and load capacity, closely matched the experimental data. This is applied to all beams that have the same geometrical features, such as size, boundary constraints, and material mechanical properties It was found that the ultimate load capacity acquired by the ABAQUS computer software was 2.75% higher than the ultimate load capacity obtained from the experiment. Present study presents a comprehensive presentation of the material properties, constraints, Schemes of loading, and interactions of the reinforced concrete beams with applied load. From the results obtained in the experiment, some of the findings consisted of load-deflection analyses and the mechanism of failure of the tested beam to check on the validity of the outcome.

Here's a list of the main concluding remarks:

1- The results obtained from the Finite Element Analysis (FEA) demonstrated a satisfactory level of agreement with the experimentally tested data concerning the general behavior, failure mechanism, load-deformation characteristics, and load-bearing capacity of the beams. This agreement is observed for beams with identical geometric parameters, including dimensions, boundary conditions (loading and support), and the mechanical properties of the materials used.

2- The experimentally determined ultimate load capacity was, on average, 2.75% lower than the values predicted by the ABAQUS software. In contrast, the difference in deflection at ultimate loads between the experimental results and numerical simulations averaged 7.54%.

8. References

1. Barbero, E. J. (2023). Finite element analysis of composite materials using Abaqus®. CRC press.
2. Pasternak, V., Ruban, A., Zolotova, N., & Suprun, O. (2023, September). Computer Modeling of Inhomogeneous Media Using the Abaqus Software Package. In Defect and Diffusion Forum (Vol. 428, pp. 47-56). Trans Tech Publications Ltd.
3. Ahmed, A. (2014). Modeling of a reinforced concrete beam subjected to impact vibration using ABAQUS. International Journal of Civil & Structural Engineering, 4(3), 227-236.
4. Guo, W., Feng, K., Zhou, Y., Lu, X., Qi, M., He, C., & Xiao, M. (2023). Experimental and numerical investigation on the shear behavior and damage mechanism of segmental joint under compression-shear load. Tunnelling and Underground Space Technology, 139, 105238.
5. Hemamathi, A., Sukumar, B., & Chantrakant, R. H. G. (2022, October). Numerical Analysis of RCC Beam Using ABAQUS. In IOP Conference Series: Earth and Environmental Science (Vol. 1084, No. 1, p. 012077). IOP Publishing.
6. Al Hasani, S., Nasrellah, H. A., & Abdulraeg, A. A. (2021). Numerical study of reinforced Concrete beam by using ABAQUS software. International Journal of Innovative Technology and Interdisciplinary Sciences, 4(3), 733-741.
7. Jadhav, A. M., & Munot, H. K. (2016). Analytical Study of Mechanism of Concrete Cracking and Its Propagation Due to Corrosion of Reinforcement in RCC. Open journal of civil engineering, 6(2), 286-294..
8. Hankare, A. V., Patil, A. N., & Deshmukh, A. R. (2014). Flexural strength of normal beam by replacing tension reinforcement as waste tyre. International Journal of Engineering

Research, 3(5), 330-332.

9. Mehra, A. S., Singh, S. B., & Kodur, V. K. R. (2022). Methodology for Predicting the Structural Response of FRP-Concrete Composite Beams Using Abaqus/CAE Software Package. In Stability and Failure of High Performance Composite Structures (pp. 417-452). Singapore: Springer Nature Singapore.

10. Rodríguez Plasencia, G., Bonilla Rocha, J. D., Hernández Santana, J. J., & Pudipedi, L. (2017). Study of the behavior of reinforced concrete deep beams. Estimate of the ultimate shear capacity. Revista de la Construcción, 16(1), 43-56.

11. Huang, Z., Chen, W., Tran, T. T., Pham, T. M., Hao, H., Chen, Z., & Elchalakani, M. (2021). Experimental and numerical study on concrete beams reinforced with Basalt FRP bars under static and impact loads. Composite Structures, 263, 113648.

12. Wani, S. B., & Mohammed, M. H. S. (2020). Experimental Study on Fractional Replacement of Cement with Waste Paper Pulp in Concrete. International Journal of Innovative Technology and Interdisciplinary Sciences, 3(4), 521-529..

13. Mishra, J., & Panigrahi, R. (2020). Mini-Review on structural performance of fiber reinforced geopolymer concrete. International Journal of Innovative Technology and Interdisciplinary Sciences, 3(2), 435-442.

14. Deng, S., Qie, Z., & Wang, L. (2015, July). Nonlinear analysis of reinforced concrete beam bending failure experimentation based on ABAQUS. In First International Conference on Information Sciences, Machinery, Materials and Energy (pp. 439-443). Atlantis Press.

15. Jaber, H. T., Sarsam, K. F., & Muhammad, B. R. (2023, March). Numerical study on the shear strength of reinforced concrete beams using ABAQUS. In AIP Conference Proceedings (Vol. 2651, No. 1). AIP Publishing.

16. Jabbar, A. M., Mohammed, D. H., & Hasan, Q. A. (2023). A numerical study to investigate shear behavior of high-strength concrete beams externally retrofitted with carbon fiber reinforced polymer sheets. International Journal of Engineering, 36(11), 2112-2123.