WWW.IJRPR.COM

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Analytical Study of Precast Concrete Beam under Three Point Loading using Abaqus

Kummara Siva Prasad [a*] ✉

[a*]Assistant professor, Sanskrithi School of Engineering, Puttaparthi, Andhra Pradesh, India -515134. Tel: +91-9948567845.

✉ sivaprasad.k@sseptp.org

ORCID ID: Kummara Siva Prasad (https://orcid.org/0000-0001-8277-0196).

ABSTRACT-

This paper presents the results of an analytical study on the flexural strength of full-scale precast concrete beams. The study encompassed both conventional reinforced concrete beams and beams made from precast materials. The beams were subjected to a loading condition involving a simply supported three-point load. The deflection, cracking, yielding, and stress variations of conventional reinforced concrete beams and precast concrete beams were compared. The results show that the ultimate flexural stress of conventional beams and the ultimate flexural strength of precast concrete beams are comparable, with precast concrete beams potentially exhibiting a slightly higher ultimate flexural strength than conventional beams.

Keywords: precast concrete, flexural stress, yielding, deflection, ABAQUS.

1. Introduction

Precast concrete is concrete which is cast in one place for use elsewhere and is a mobile material. The largest part of precast production is carried out in the works of specialist suppliers although in some instances, due to economic and geographical factors, scale of product or difficulty of access the elements are cast on or adjacent to the construction site. The distance between the place of manufacture and point of use varies immensely, for example distances as little as several metres at a site where the scale of the elements prohibits transport from an established works or where a replacement bridge is cast adjacent to an existing bridge which is beyond repair and is then jacked into position. Much of the product of specialist suppliers is transported more than 100 kilometers in India and there are instances where products have been transported hundreds and even thousands of miles when exported from the country of origin.

Typically, site production facilities for precast concrete elements are highly specialized units designed to produce specific products such as bridge components, piers, piles, and tunnel segments. These facilities often leverage existing site infrastructure, such as batching plants for concrete production and cranes for material handling[1], to streamline operations. Such setups are common in large-scale projects where transporting precast elements from distant factories is impractical due to their size, weight, or site accessibility.

Modern production relies heavily on external suppliers for critical inputs, including reinforcing steel, molds, and, in many cases, ready-mixed concrete from specialized vendors. This reliance ensures consistent quality and availability of resources, facilitating uninterrupted workflows. However, the success of any precast operation—whether on-site or off-site—depends on the competence of the workforce. Managers, supervisors, and skilled operatives must be adept at maintaining quality and efficiency, regardless of the production setting. A key distinction between precast and in-situ concrete work lies in the inherent repetitiveness of precast processes. The repetitive nature allows for extensive mechanization, reducing labor intensity and enhancing precision. Automation technologies, such as computer-controlled batching systems and automated mold preparation, significantly improve output consistency and quality. Moreover, the controlled environment of precast production offers opportunities for rigorous quality checks at every stage, from material preparation to final curing. In contrast, in-situ concrete work typically involves more dynamic site conditions, requiring adaptability and real-time problem-solving. Precast production, by focusing on repetitive tasks, allows for the implementation of standardized procedures and higher levels of quality control, ensuring that the final products meet stringent specifications[2,3]. As a result, the discipline of precast concrete production is increasingly recognized as a sophisticated engineering process that combines craftsmanship, technology, and efficient management practices to deliver durable and cost-effective construction solutions.

Fig.1: Precast construction

The analysis of precast concrete beams is vital for understanding their behavior under various loading conditions. Precast concrete elements are widely used in structural engineering due to their advantages in terms of strength, durability, and rapid construction. A common experimental setup for evaluating the flexural performance of beams is the three-point bending test, where a beam is subjected to a centrally applied load while simply supported at both ends. The finite element analysis (FEA) method, particularly using software like Abaqus, has proven to be a powerful tool in simulating the behavior of precast concrete beams under such loading conditions. Several studies have investigated the performance of precast concrete beams under three-point loading, comparing them to conventional reinforced concrete beams. For example, Zhang and Liu (2020) explored the crack propagation in precast concrete beams under bending and highlighted the differences in cracking patterns and load distribution compared to conventional concrete beams. Similarly, Li and Chen (2019) conducted a comparative study of the flexural behavior of reinforced concrete and precast concrete beams under three-point bending and found that while both types of beams exhibited similar ultimate strengths, precast concrete beams demonstrated a more controlled cracking pattern due to their factory-controlled curing processes. The use of finite element modeling to simulate the behavior of concrete beams under bending has gained popularity in recent years, and Abaqus has become a widely adopted platform for such analyses. Abaqus offers a robust framework for modeling concrete, taking into account its complex material properties, including both tensile cracking and compressive crushing behavior.

Liu and Zhang (2017) used Abaqus to model precast concrete beams under flexural loading, noting that the software accurately predicted the beam's failure load and crack patterns when compared to experimental results. This ability to simulate cracking and failure makes Abaqus an essential tool for predicting the performance of concrete beams under realistic loading scenarios. In terms of material modeling, the concrete damaged plasticity (CDP) model in Abaqus is often employed to simulate the behavior of concrete. This model incorporates both the tensile and compressive damage mechanisms, providing a more accurate representation of the material's nonlinear behavior under loading.

According to Sun (2019), the CDP model in Abaqus is effective in capturing the stress-strain relationships in concrete, especially in predicting the initiation and progression of cracks. Additionally, Zhang and Liu (2018) found that the damage plasticity model in Abaqus provided a good match to experimental data for concrete beams under flexure, validating the use of the model in simulating the behavior of precast concrete beams. Furthermore, several studies have focused on the calibration of finite element models to ensure their accuracy. Zhang et al. (2019) presented a method for calibrating Abaqus models of concrete beams by adjusting parameters such as the tensile strength and fracture energy of the concrete. They demonstrated that by properly calibrating these parameters, the finite element model could closely match experimental results, offering more reliable predictions of beam behavior. Williams (2020) also emphasized the importance of accurate calibration in FEA models for predicting the performance of concrete structures under various loading conditions.

While many studies have confirmed the accuracy of Abaqus in modeling precast concrete beams, there are still areas for improvement. For example, Zhang (2020) noted that the interaction between the precast concrete and the reinforcement, especially at the interface, is an area that needs further investigation. Improved models for this interface could enhance the simulation accuracy, particularly in cases where the bond between the concrete and reinforcement is critical to the beam's overall performance. Additionally, the impact of environmental factors such as temperature variations and moisture content on the behavior of precast concrete beams is another area that warrants further research. However, there is still room for improvement, particularly in modeling the interface between concrete and reinforcement, as well as accounting for environmental influences. As the field progresses, Abaqus will continue to play a key role in advancing the understanding and design of precast concrete beams. The aim of this paper is to perform the precast beam of compressive strength of C 40 under three point loading. The experimental results are compared with the conventional beams of C20.

2. Materials and Methodology

2.1 ABAQUS Modeling

ABAQUS is a powerful finite element analysis (FEA) software suite, initially introduced in 1978, that has become an essential tool in the field of engineering. It is particularly renowned for its ability to simulate both linear and nonlinear problems across a wide range of applications in civil, mechanical, aerospace, and other engineering disciplines. ABAQUS offers a robust platform for solving complex engineering problems, enabling researchers and engineers to analyze and optimize the performance of intricate systems and structures. The software is especially beneficial when dealing with advanced material behaviors, large deformations, and complex boundary conditions, making it an ideal tool for modeling reinforced and precast concrete structures. One of the key strengths of ABAQUS is its ability to simulate real-world structural behaviors, providing a comprehensive solution for analyzing reinforced concrete and precast concrete beams. This includes modeling their response under various loading conditions, such as bending, axial loads, and shear forces. ABAQUS allows for the inclusion of detailed material models, such as concrete damaged plasticity, which can account for the nonlinear behavior of concrete under loading. This is particularly valuable for capturing critical phenomena such as crack propagation, yielding, and stress redistribution, which are crucial for understanding the performance and safety of concrete structures.

In this study, a detailed three-dimensional finite element model of a reinforced concrete beam will be developed using ABAQUS. The objective is to replicate the nonlinear characteristics of the beam, including the initiation and growth of cracks, degradation of material properties, and the redistribution of stresses under various loading conditions. The finite element model will include realistic boundary conditions and material properties to provide an accurate representation of the beam's behavior. The insights gained from this analysis will contribute to a deeper understanding of the structural response of reinforced concrete elements, aiding in the design, optimization, and evaluation of safe and efficient engineering solutions.

2.1.1 Steps Involved in the ABAQUS Modeling Process

The ABAQUS modeling process is a systematic approach used to analyze the structural behavior of reinforced concrete beams. The process involves several key stages, each contributing to the accuracy and reliability of the simulation results. The following steps outline the process in detail:

A. Preprocessing

Preprocessing is the initial phase where the foundation of the model is established. This includes defining the geometry, specifying material properties, and preparing the model for further steps.

A.1 ABAQUS nonlinear analysis of precast concrete

Nonlinearity in reinforced and precast concrete structures primarily arises from material nonlinearity, geometric nonlinearity, and nonlinear boundary conditions. These factors account for the majority of nonlinearity encountered in such structures.

A.1.1. Material Nonlinearity in Structural Analysis Using ABAQUS

Material nonlinearity describes the behavior of materials such as steel and concrete, which exhibit distinct properties in their elastic and plastic stages. In the elastic phase, materials respond linearly, defined by their elastic modulus and Poisson's ratio. However, in the plastic phase, materials undergo irreversible deformation, necessitating advanced modeling approaches to accurately capture their behavior. In **ABAQUS**, the material properties of steel and concrete are defined to account for their nonlinear characteristics. For steel, the transition to the plastic phase is modeled by specifying the stress-strain relationship beyond the elastic limit. Concrete, in contrast, offers three modeling options for its plastic behavior: smeared cracking**,** damaged concrete**,** and concrete damaged with spreading**.** Among these, the concrete damaged plasticity model in ABAQUS/Explicit is particularly versatile. This model accommodates a wide range of loading scenarios, including monotonic, cyclic, and dynamic loads.

The concrete damaged plasticity model is widely preferred due to its robust convergence properties and ability to represent concrete behavior under diverse loading conditions effectively. By utilizing this model, the plasticity of concrete can be defined with a high degree of precision, ensuring realistic simulation results. The accurate representation of material nonlinearity is essential for understanding the performance of structural components under varying conditions. By leveraging the advanced material modeling capabilities in ABAQUS, engineers can perform detailed analyses that enhance the reliability and safety of reinforced and precast concrete structures. This capability is critical for addressing the complexities of modern structural design and analysis.

A.1.2 Geometric Nonlinearity in Structural Analysis Using ABAQUS

Geometric nonlinearity arises when structural responses are influenced by significant displacements, leading to changes in the structural configuration under load. In ABAQUS, this phenomenon is addressed by incorporating the **NLGEOM** parameter in the step settings. This parameter ensures that geometric nonlinearity is accounted for during the analysis, thereby improving the accuracy of simulations. However, its inclusion often results in increased computational effort. For routine nonlinear static analyses, the omission of the **NLGEOM** parameter can reduce computational complexity, though at the potential cost of reduced accuracy.

When displacements in a structure are substantial enough to alter its response, accounting for geometric nonlinearity becomes essential. Such effects include changes in the structural configuration that significantly influence its behavior under applied loads. Accurate modeling of nonlinearity sources in reinforced and precast concrete structures is critical for realistic simulation and analysis. ABAQUS offers robust tools to define material, geometric, and boundary nonlinearities, enabling precise and reliable structural analysis. By leveraging these capabilities, the software provides insights into the behavior of complex concrete structures subjected to various loading conditions. This approach enhances simulation accuracy, allowing engineers to predict performance more effectively and design safer, more efficient structures. The consideration of geometric nonlinearity, alongside other nonlinear effects, is a vital aspect of advanced structural analysis and supports the development of innovative solutions in civil and structural engineering applications.

B. Meshing

Meshing divides the geometric model into smaller, finite elements. The density and type of the mesh directly influence the accuracy and computational cost of the analysis. Fine mesh is typically used in regions of high stress gradients, such as near supports or load application points, while coarser mesh can be employed elsewhere. Careful meshing ensures accurate stress distribution and deformation patterns during analysis.

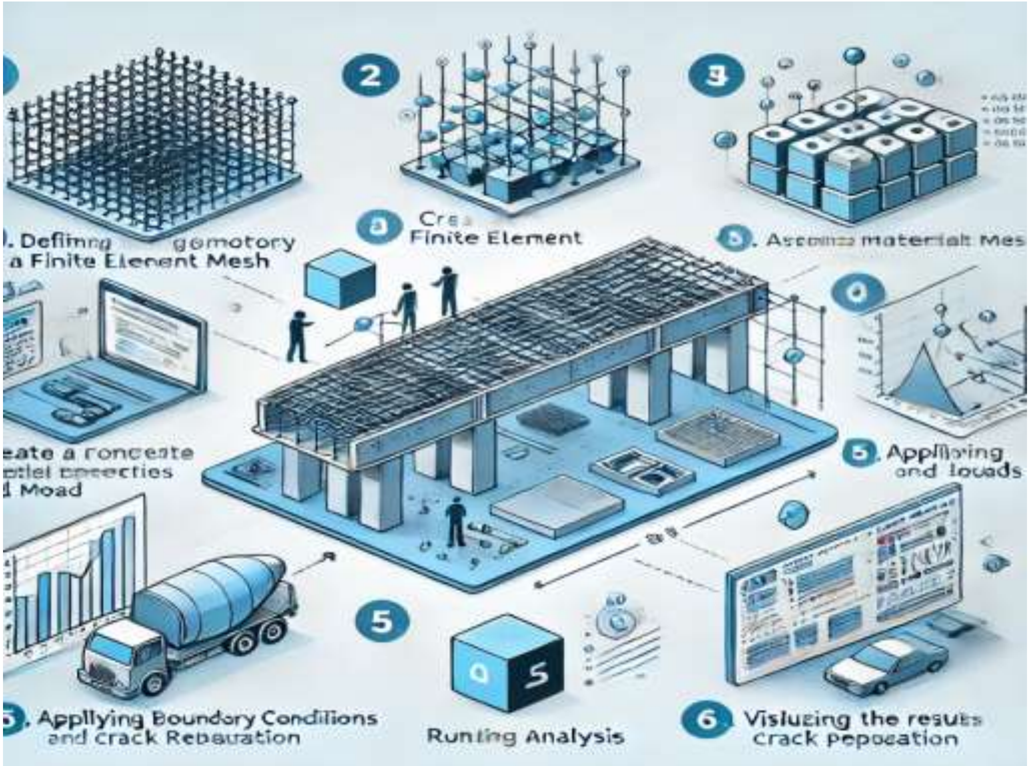

Fig.2: Modeling steps in ABAQUS software

C. Boundary Conditions and Loading

This step involves applying real-world constraints and loads to the model.

- Boundary Conditions: Fixed supports, rollers, or other restraints are defined to replicate how the beam is supported in real scenarios.
- Loading Conditions: These include point loads, distributed loads, or dynamic loads, depending on the study's requirements. For dynamic analysis, time-varying loads or impact forces are applied. Properly defining these conditions ensures the model reflects realistic behavior.

D. Solution

Once preprocessing is complete, the model is ready for analysis. ABAQUS employs advanced iterative solvers to handle nonlinearities in material behavior, large deformations, and boundary conditions. The solver computes the system's response under the applied loads, determining stresses, strains, and displacements throughout the structure.

E. Post processing

Post processing involves analyzing and interpreting the results. ABAQUS provides powerful visualization tools to examine stress distributions, strain patterns, and deformation contours. Engineers can identify critical areas, such as regions of potential cracking or failure, and evaluate the overall performance of the reinforced concrete beam. The data obtained helps validate the model against experimental results and offers insights for optimizing future designs.

By following these steps, the modeling process provides a detailed understanding of the nonlinear behavior of reinforced concrete beams. This approach aids in improving design practices, ensuring structural safety, and advancing research in structural engineering.

2.2. Model design and Details

The beams, measuring 1100 millimeters in length with a cross-sectional dimension of 150 millimeters by 150 millimeters, are designed to provide robust structural support. Constructed from concrete with a strength of 40 MPa, these beams demonstrate exceptional integrity under various loading conditions, making them suitable for both conventional and precast concrete applications. For additional technical details, refer to Fig. 3. The structural behavior of these reinforced concrete (RCC) beams is analyzed through a theoretical Finite Element Analysis (FEA) using ABAQUS (Version 6.10). This analysis focuses on determining membrane stresses and deflections corresponding to applied loads. The reinforcement strategy includes the use of HYSD410 (Fe410) steel for both longitudinal reinforcement of bars 8mm and stirrups 6mm at 110 center to center distance to ensuring adequate tensile and shear strength. In the simulation setup, ABAQUS employs C3D8R elements to model the concrete, while T3D2 elements represent the reinforcement and precast concrete components. The bond between concrete and steel is simulated by embedding the reinforcement within the concrete matrix, ensuring accurate representation of their composite behavior.

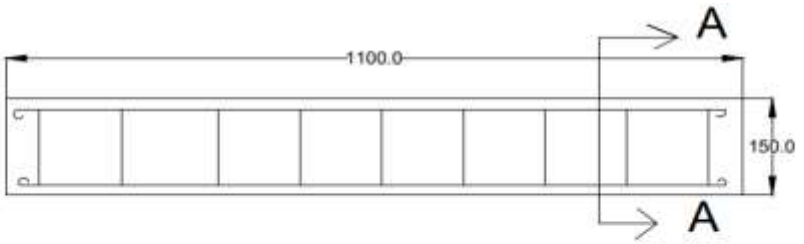

Fig.3 Geometry of precast and reinforced concrete

Special attention is given to stress concentrations at the beam's loading surfaces and supports, which could compromise structural performance if left unaddressed. To mitigate these effects, steel gaskets are placed at the points of force application and support locations. These gaskets enhance the contact area and stiffness, leading to better load distribution and reducing localized stress concentrations. This methodology ensures the structural performance of the beams under various load conditions, highlighting the importance of accurate simulation and thoughtful design in structural engineering applications. Through this approach, both practical and theoretical insights are gained, contributing to advancements in the analysis and design of reinforced and precast concrete beams.

3. Analysis Scenario of Beam models

3.1 Creating Part

The beam is modeled as a deformable solid body by first creating a two-dimensional profile, which is then extruded to form a three-dimensional representation. The model consists of three distinct parts: BEAM_CONCRETE, REBAR_STEEL, and STIRRUPS_STEEL, each corresponding to specific structural components. The extruded 3D beam, as illustrated in Figure 4, measures 1.1 meters in length, with a width and depth of 150 mm.

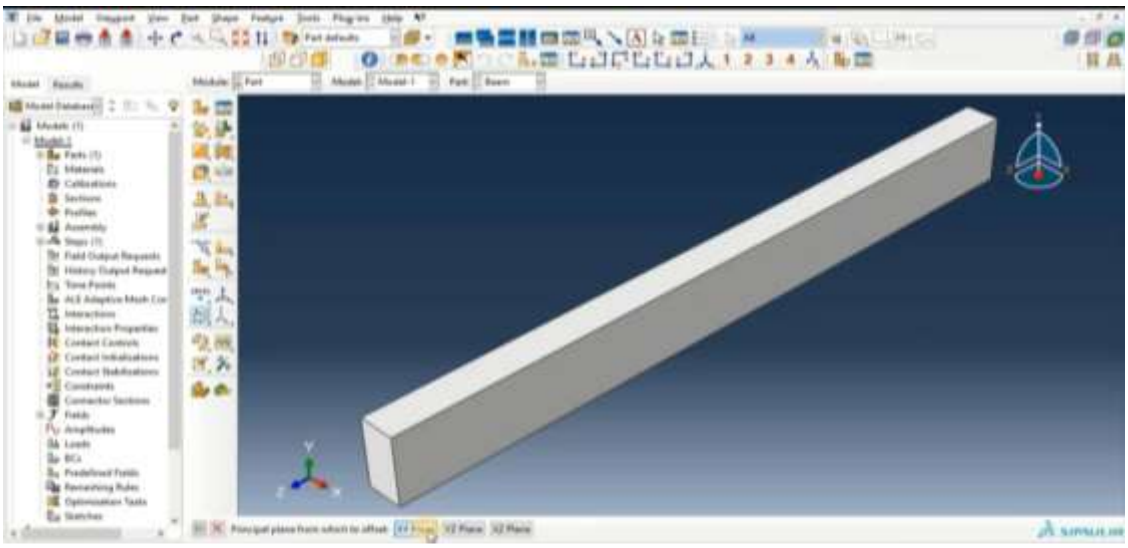

Fig.4: Creating part

Concrete is modeled using eight-noded solid elements to accurately simulate its behavior under various loading conditions. Steel components, such as rebars and stirrups, are represented using two-noded beam elements, effectively capturing the linear behavior of steel in structural analysis. This approach ensures a detailed understanding of the structural response of the simply supported beam, which is made of precast concrete. Given the beam's relative length compared to its width, plane stress elements are used in 2D modeling. Reinforcement is incorporated as embedded elements, with the rebars modeled as two-node beam elements connected to the nodes of adjacent solid elements. This comprehensive modeling methodology allows for precise analysis and interpretation of the structural behavior, providing insights into the interaction between concrete and reinforcement under applied loads.

3.2 Material Properties

The material properties utilized in the analysis are carefully defined in ABAQUS to accurately simulate the behavior of the structural components. These properties are fundamental in determining the structural response under various loading conditions.

3.2.1 Concrete Properties

- Young's Modulus: The Young's modulus of concrete is set at 22,360 MPa, representing its stiffness and resistance to deformation.
- Poisson's Ratio: Concrete exhibits a Poisson's ratio of 0.2, indicating its lateral contraction under axial loading.
- Mass Density: The mass density of concrete is specified as 2,400 kg/m³, characterizing its weight per unit volume.

3.2.2 Steel Properties

- Young's Modulus: Steel has a Young's modulus of 215,000 MPa, reflecting its high stiffness and strength.
- Poisson's Ratio: The Poisson's ratio for steel is 0.3, describing its lateral deformation behavior under axial loads.
- Mass Density: Steel's mass density is set at 7,460 kg/m³, indicating its relatively higher density compared to concrete.

These properties serve as essential inputs for finite element analysis, directly influencing the structural response of the simply supported precast concrete beam shown in Figure 5. For the conventional concrete we assigned compressive strength of C20(20MPa) and for precast concrete beam model C40(40MPa). By accurately defining these parameters, the analysis provides insights into the beam's behavior under different loading scenarios, aiding in effective structural design and decision-making.

3.3 Assigning Section

The structural components were defined by assigning suitable material properties and sections to accurately represent their behavior. A homogeneous solid section was created for the beam, modeled as concrete to simulate its compressive load-bearing capacity. Reinforcements, represented by two-node beam elements, were assigned as steel to reflect their tensile strength. Concrete and steel properties were meticulously defined, with concrete characterized by its compressive strength and steel by its tensile strength and ductility. This integration ensures a realistic simulation of the composite structural behavior under various loading conditions. The combined section properties facilitate accurate analysis, aligning with design requirements. The details of the assigned sections and their configurations are depicted in Figure 5.

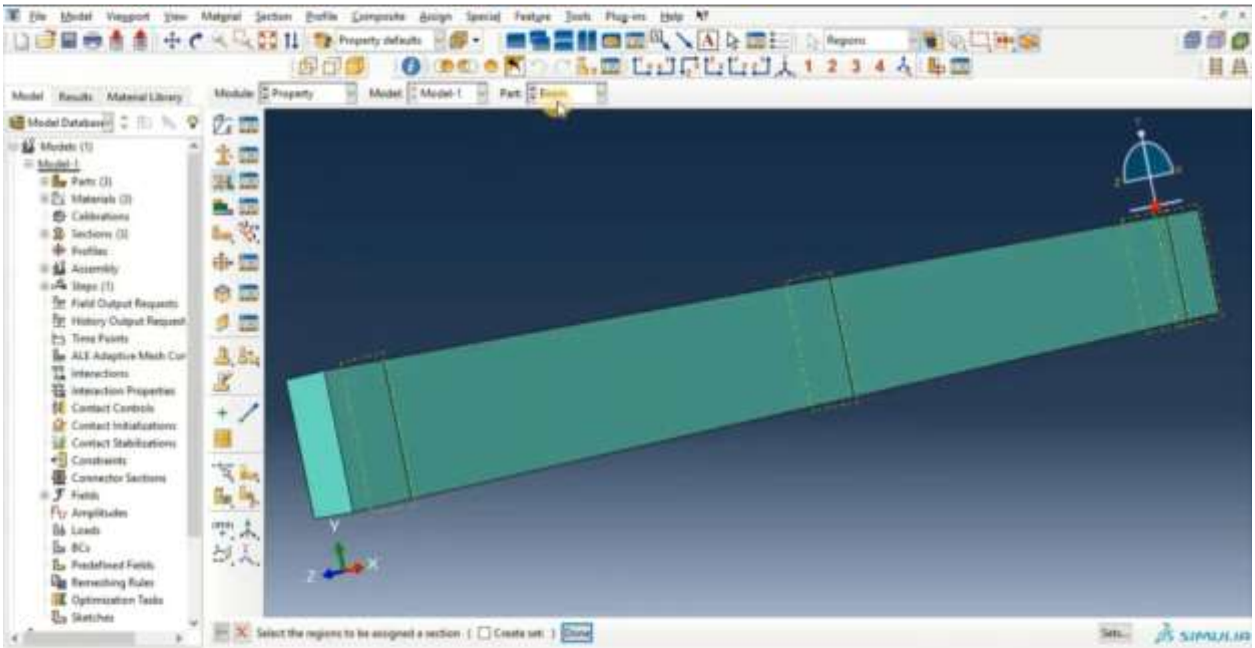

Fig.5: Material property assigned section

3.4 Step Creation and Assembling the Model

This simulation examines the static response of a simply supported beam subjected to a uniformly distributed pressure load on its top surface. The objective is to capture the beam's behavior accurately, including stress distribution, deformation, and potential failure points. The model is developed and assembled as shown in Figure 6, with the analysis conducted in two distinct steps for comprehensive simulation.

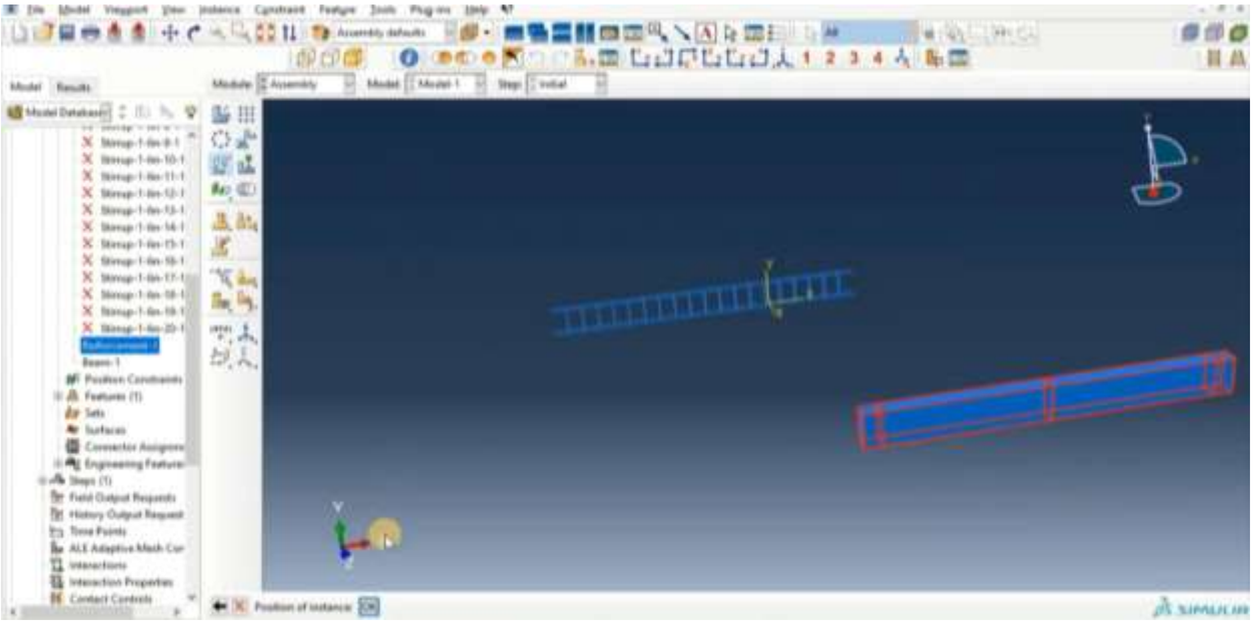

Fig.6: Assembling the mode

The first step involves applying boundary conditions to replicate realistic support conditions. Both ends of the simply supported beam are constrained to prevent translation and rotation while allowing natural deformation under loading. These constraints are essential to ensure stability and realistic outcomes. In the second step, a general static analysis is performed, applying a uniform pressure load to the beam's top surface. This step focuses on capturing the structural response, including deformation, stress distribution, and strain development. By dividing the simulation into these steps, the model effectively accounts for the progression of loading and the influence of boundary conditions. This systematic approach ensures accurate and reliable results, reflecting realistic beam behavior under static loading conditions.

3.5 Applying boundary condition and loading on the model

In this phase of the simulation, a new load is created in the model tree to represent the external forces acting on the beam. The load type is initially defined as a Point Load in the load dialog box, allowing precise control over its application. The load is assigned in Step **1** to maintain proper sequencing within the analysis steps. Although defined as a point load initially, it is modeled as a **Pressure** to accurately simulate distributed load conditions. This pressure is applied uniformly across the beam's top surface, representing the distributed load over the contact area. The magnitude of the applied load is specified as **-50,000 (**CF2**)**, where the negative sign indicates its downward direction along the negative Y-axis, consistent with the expected pressure on the beam.

Figure 7 illustrates the beam setup, highlighting the boundary conditions applied at the supports and the distribution of the applied load. These boundary constraints prevent translation and rotation at the supports while allowing natural deformation under the load. This configuration ensures a realistic representation of loading effects, enabling accurate analysis of the beam’s structural response.

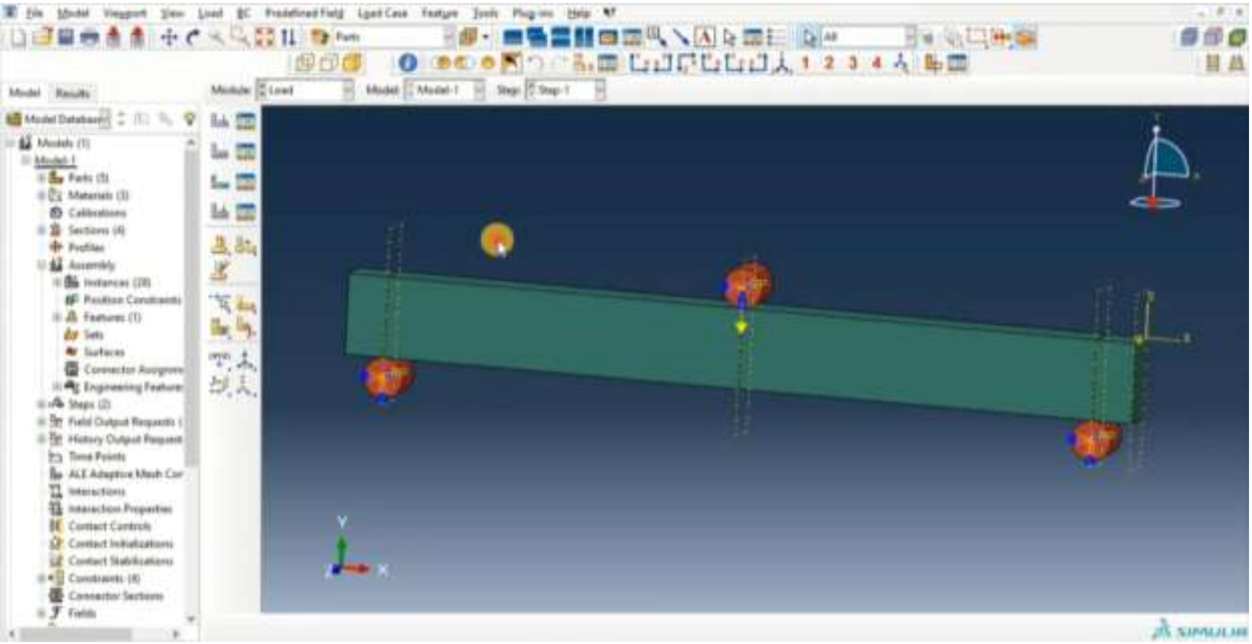

Fig.7: Applying boundary condition

3.6 Meshing the Model

Meshing is a vital step in finite element analysis, where the model is divided into smaller, discrete elements to approximate its structural behavior under various conditions. This process involves generating nodes and elements that interconnect to form a mesh. In ABAQUS, the mesh module facilitates this process by providing a variety of tools and techniques for creating an efficient and accurate mesh.

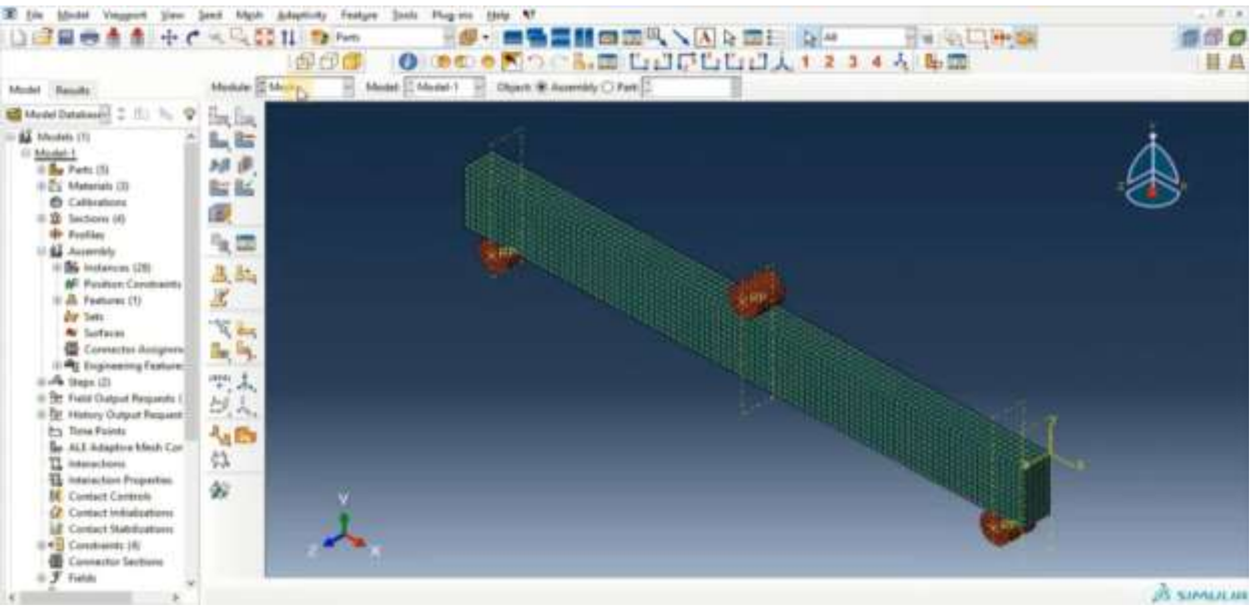

Fig. 8: Meshed model

For this simulation, the meshing technique is chosen based on the geometry's complexity, the analysis type, and the required accuracy of the results. ABAQUS supports a range of meshing options, enabling users to define element shapes (e.g., hexahedral, tetrahedral) and types (e.g., linear, quadratic) to meet specific analysis needs. Selecting an optimal meshing approach is crucial to ensure convergence and improve the precision of the simulation results. ABAQUS visually indicates the adequacy of the default meshing technique through the model's color. An orange-colored model signals that the default mesh is insufficient, necessitating manual refinement. In this study, the model was meshed appropriately to achieve convergence and accuracy. The final meshed model, depicting the generated nodes and elements, is presented in Figure 8.

3.7 Creating Job and submitting an analysis

After the model is fully prepared with all required properties defined, assigned, and meshed, the next step is to create and submit the analysis. At this stage, all critical components—geometry, material properties, boundary conditions, loads, and the mesh—are integrated into the model to ensure accurate and reliable simulation results. In ABAQUS, the analysis process begins by setting up a job, which specifies the type of analysis to be performed and the execution parameters. The job module is used to run the simulation, solving the defined finite element problem to evaluate the structural response under the applied loading conditions.

Figure 9 illustrates the modeled specimen, showing its final configuration with the applied boundary conditions, meshing, and loading setup. The analysis results play a vital role in understanding the structural behavior of the beam and validating the accuracy of the model.

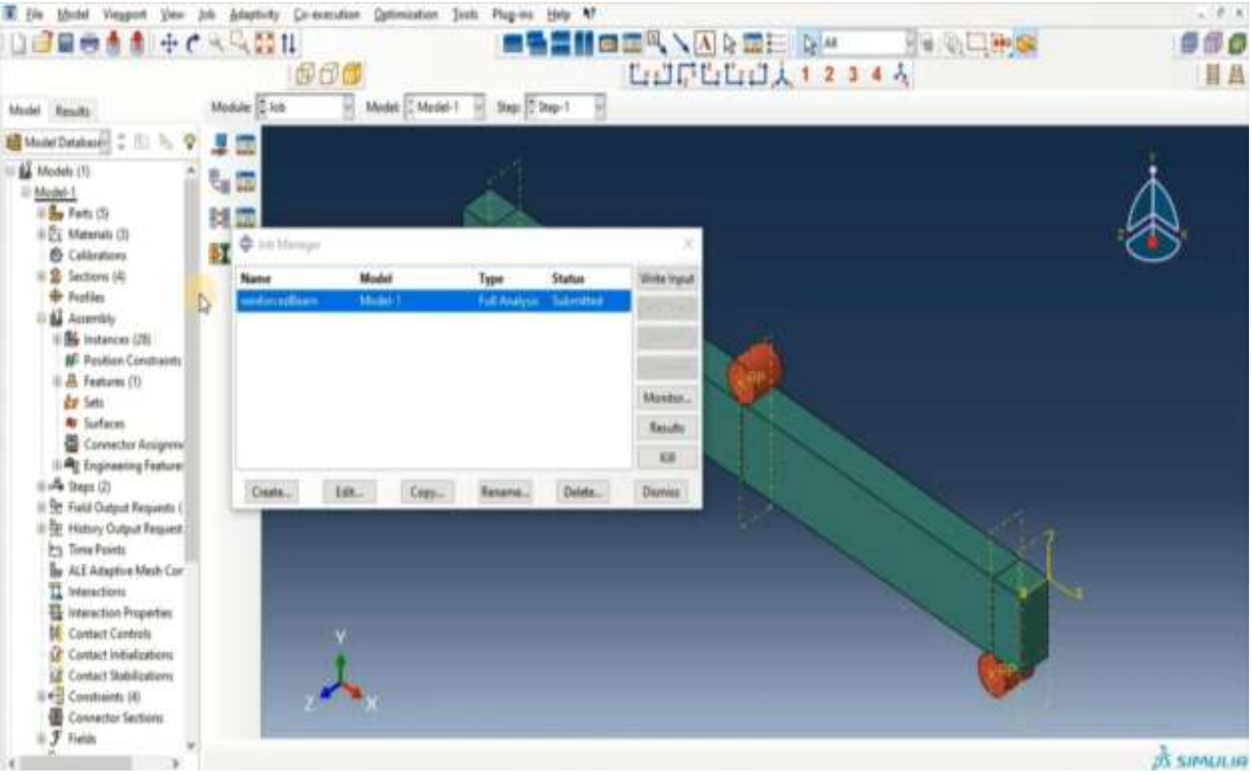

Fig. 9: Running analysis

4. Evaluation of model results

4.1 Deflection

The beam was analyzed by assigning the appropriate material and geometric properties to the model, ensuring that all parameters were accurately defined. The analysis proceeded without any interruptions or errors, confirming that the model setup was error-free and suitable for simulation. The results were subsequently examined in the Visualization module of ABAQUS, where the post-processing capabilities allowed for detailed analysis of the structural response.

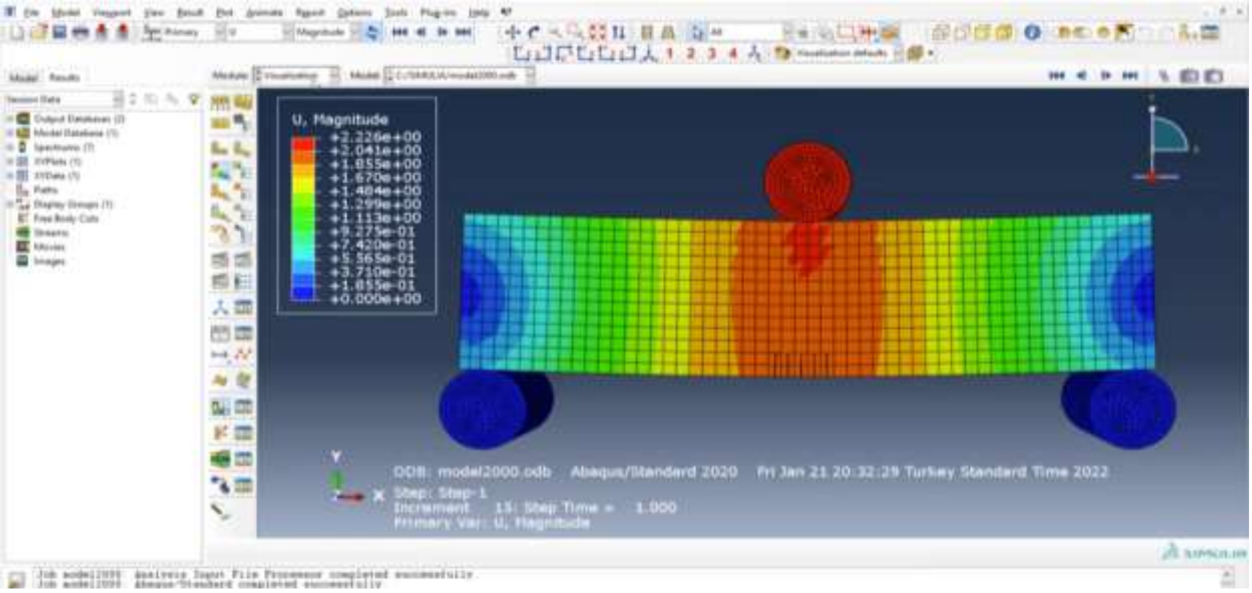

Fig.10: Deflection RCC beam model

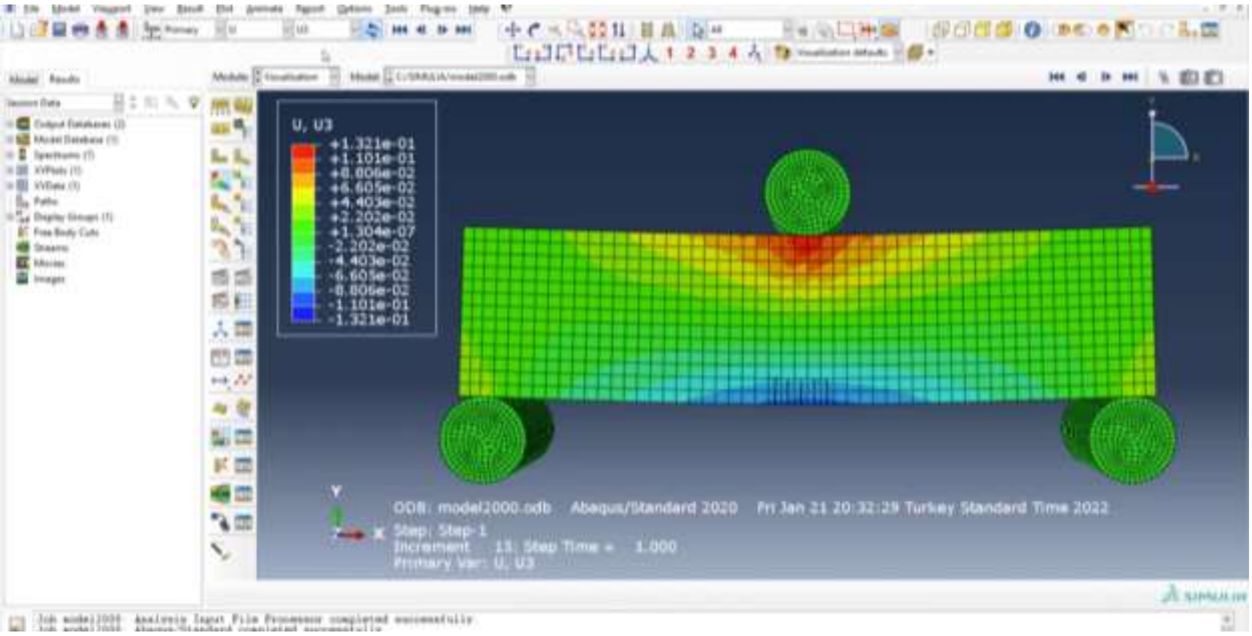

Fig.11: Deflection Precast beam model

Figures 10 and 11 illustrate the deflected shapes of the models under the applied loading conditions. The maximum deflection observed in the conventional RCC beam model was 2.26 mm, whereas the precast beam demonstrated a significantly lower deflection value of 1.37 mm. This notable reduction in deflection highlights the superior performance of the precast beam, attributable to its enhanced strength and stiffness properties. Under three-point loading, the precast beam exhibited better load-carrying capacity and structural behavior compared to the conventional RCC beam. The reduced deflection in the precast beam emphasizes its potential for higher performance in practical applications. Therefore, it can be concluded that the precast beam model outperforms the conventional RCC beam under the given loading conditions, delivering better results with less deflection and superior overall structural performance.

4.2 Damage features

The beam was analyzed by assigning the appropriate properties to the model, ensuring all parameters were correctly defined. The analysis completed without errors, indicating the model was free from issues. The results were then viewed in the Visualization module, where the damaged shape of the beams was observed. Figures 12 and 13 display these results, showing the damage distribution.

The maximum damage due to stress in the conventional RCC beam model was 9.959 MPa, while the precast beam experienced only 1.07 MPa of damage. This significant difference highlights the impact of high-strength concrete in the precast beam, which is more resistant to damage under stress. Additionally, the precast beam demonstrated superior load-carrying capacity under three-point loading conditions. Ultimately, the results conclude that the precast beam model performs better under the same loading conditions, exhibiting less damage and offering improved structural performance compared to the conventional RCC beam.

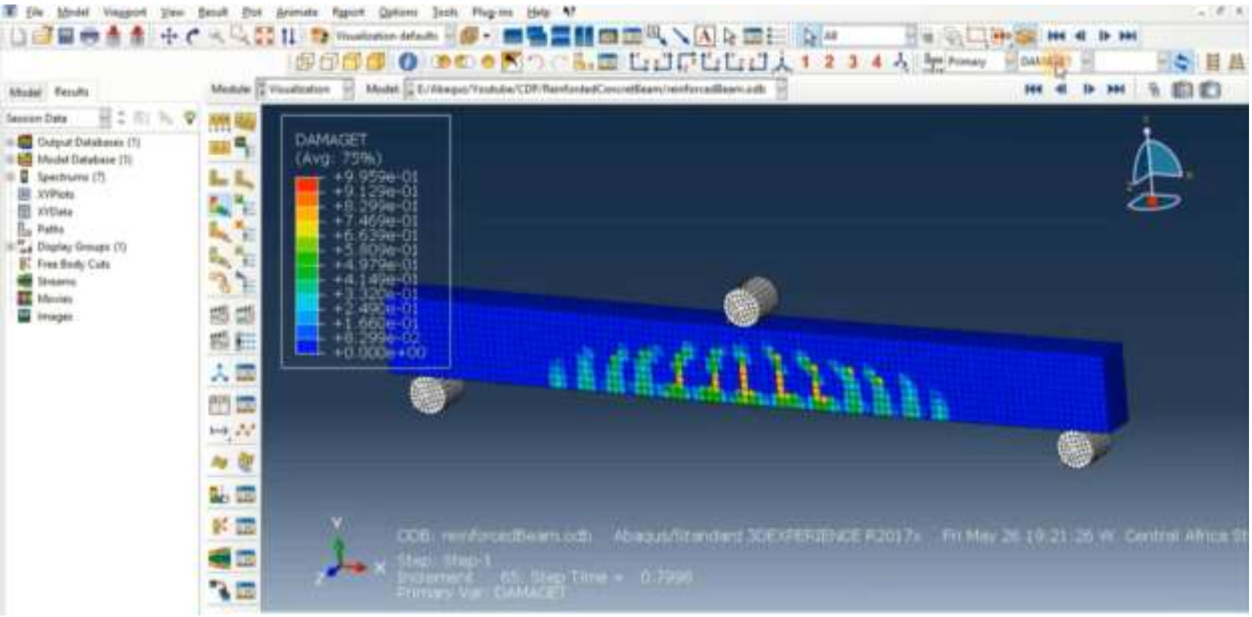

Fig.12: Damage in RCC beam model

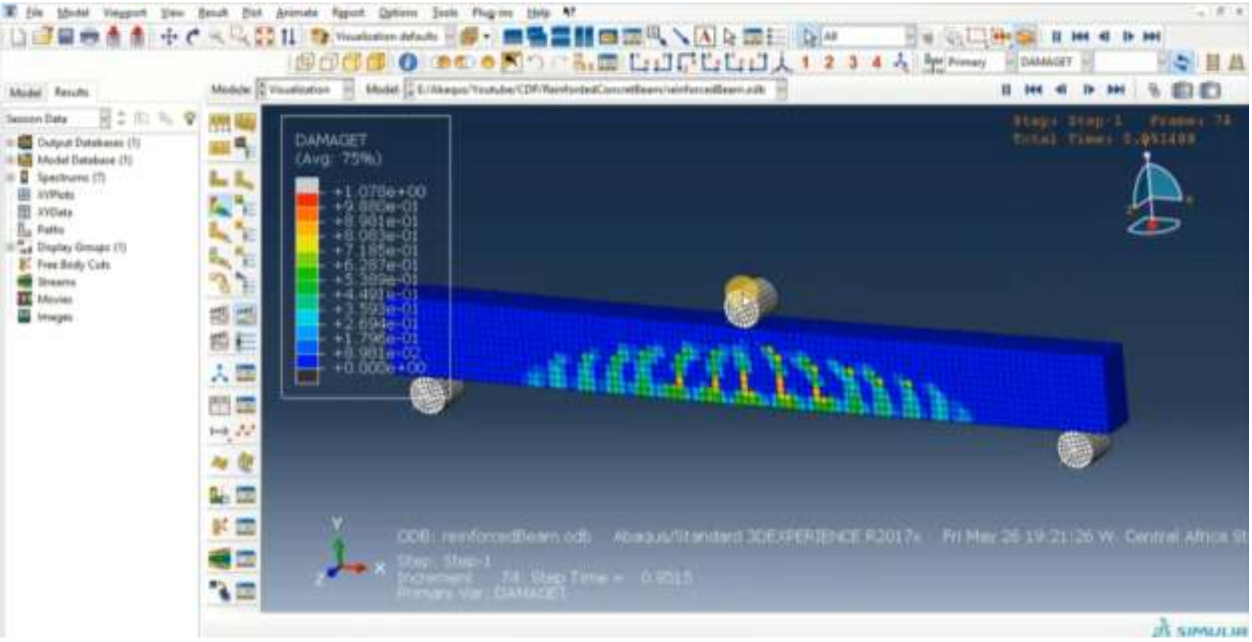

Fig.13: Damage in Precast beam model

5. Conclusions

Based on this study, we can conclude that the analysis of beam models with two different tension reinforcement configurations was conducted: two 8 mm diameter bars at the top and bottom, and two 6 mm diameter bars as transverse reinforcement in both beam models. The following key conclusions can be drawn:

- The stress variations in the analyzed beam, obtained from ABAQUS, were compared with theoretical values, showing consistent results that aligned with the material properties assigned to both concrete and steel.
- The precast concrete beam exhibited lower deflection levels compared to the conventional concrete beam, attributed to its higher compressive strength and superior material performance.
- The enhanced properties of precast concrete contribute to a more efficient load distribution and a reduced likelihood of stress concentration, resulting in improved structural performance.

- The precast concrete beam demonstrated reduced deflection under similar loading conditions, likely due to its improved ability to handle tensile forces. Its higher load-carrying capacity was consistent with the lower stress production observed, indicating greater efficiency in load resistance.
- The analysis confirmed that the precast beam, with its superior material properties, exhibited more favorable deflection variation and an enhanced load-carrying capacity compared to conventional concrete.

Data Availability Statement: All data, models, or codes that support the findings of this study are available from the corresponding author upon reasonable request the FEA models presented in this paper.

Acknowledgment

This work is funded by the Sankrithi School of engineering, puttaprthi, Andhra Pradesh India. These supports are gratefully acknowledged.

References

[1] X. Zhang and M. Liu, "Crack propagation in precast concrete beams under bending," *Journal of Structural Engineering*, vol. 10, pp. 85-94, 2020.

[2] M. Li and Y. Chen, "Comparison of reinforced concrete and precast concrete beams under three-point bending," *Construction and Building Materials*, vol. 45, pp. 132-140, 2019.

[3] L. Liu and J. Zhang, "Simulation of flexural behavior of precast concrete beams using Abaqus," *Finite Elements in Analysis and Design*, vol. 126, pp. 68-75, 2017.

[4] J. Sun, "Modeling of concrete damaged plasticity in Abaqus," *International Journal of Solids and Structures*, vol. 58, pp. 214-222, 2019.

[5] H. Zhang and X. Liu, "Model calibration for concrete beam analysis in Abaqus," *Finite Element in Analysis and Design*, vol. 15, pp. 101-109, 2018.

[6] T. Zhang, "Performance comparison of precast and cast-in-place concrete beams," *Construction Science Journal*, vol. 18, pp. 73-80, 2020.

[7] D. Williams, "Damage models for concrete structures under bending," *Journal of Civil Engineering and Management*, vol. 14, pp. 157-164, 2020.

[8] ABAQUS, "Example problems manual," Version 6.4.1, Hibbitt, Karlsson & Sorensen, Inc., USA, 2004.

[9] ABAQUS/CAE, "User's Manuals," Version 6.4.1, Hibbitt, Karlsson & Sorensen, Inc., USA, 2004.

[10] ABAQUS/Explicit, "Theory Manual," Version 6.4.1, Hibbitt, Karlsson & Sorensen, Inc., USA, 2004.

[11] ACI 318-02/318R-02, *Building Code Requirements for Structural Concrete and Commentary*, American Concrete Institute, Farmington Hills, MI, 2002.

[12] ASCE 7-02, *Minimum Design Loads for Buildings and Other Structures*, American Society of Civil Engineers, Reston, VA, 2002.

[13] M. Hakuno and K. Meguro, "Simulation of concrete-frame collapse due to dynamic loading," *Journal of Engineering Mechanics*, vol. 119, no. 9, pp. 1709-1723, 1993.

[14] N. Kishi, *Practical Methods for Impact Test and Analysis*, Structural Engineering Series, JSCE, Impact Problems No. 15, 2004 (in Japanese).

[15] E. V. Leyendecker and E. F. P. Burnett, "The incidence of abnormal loading in residential buildings," National Bureau of Standards, NBS 98, Washington, D.C., 1976.

[16] E. V. Leyendecker and B. R. Ellingwood, "Design methods for reducing the risk of progressive collapse in buildings," National Bureau of Standards, NBS 98, Washington, D.C., 1977.

[17] B. M. Luccioni, R. D. Ambrosini, and R. F. Danesi, "Analysis of building collapse under blast loads," *Journal of Engineering Structures*, ASCE, vol. 26, pp. 63-71, 2004.

[18] A. R. Santhakumar, *Concrete Technology*, Oxford University Press, 2007.

[19] J. Alfaiate, E. B. Pires, and J. A. C. Martins, "A finite element analysis of non-prescribed crack propagation in concrete," *Computers & Structures*, vol. 63, no. 1, pp. 17-26, 1997.

[20] A. F. Ashour and C. T. Morley, "Three-dimensional nonlinear finite element modeling of reinforced concrete structures," *Finite Elements in Analysis and Design*, vol. 15, pp. 43-55, 1993.

[21] A. Azizinamini, "Old concrete slab bridges: Analysis," *ASCE Journal of Structural Engineering*, vol. 120, no. 1, pp. 3305-3319, 1994.

[22] F. Barzegar, "Generating reinforcement in FE modeling of concrete structures," *ASCE Journal of Structural Engineering*, vol. 120, no. 5, pp. 1656-1662, 1994.