International Journal of Research Publication and Reviews

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

Nonlinear Analysis of precast Concrete Beam Bending Failure Experimentation Based on ABAQUS

Kummara Siva Prasad *

Research Scholar, School of Civil Engineering, Vellore Institute of Technology, Vellore-632014, Tamil Nadu, India. Tel: +91-9948567845.
kumarasiva.prasad2019@vitstudent.ac.in

ABSTRACT –

The nonlinear analysis of reinforced concrete beam and precast concrete beam were conducted based on the finite element analysis software ABAQUS. In this simply supported beam analysis, the plasticity model of concrete damage in ABAQUS has been introduced thoroughly. Finally, the results of the ABAQUS analysis were compared of precast and reinforced concrete beam in a diagram, accordingly reasons of the result difference between the two beams were discussed, which can be a useful reference for the further study of the nonlinear analysis of precast concrete beams.

Keywords: Nonlinear analysis; Reinforced concrete beam; precast beam bending failure experimentation; Finite element analysis.

1. Introduction

The development of finite element analysis software is developing alongside the ongoing development of finite element theory and computer technology. The use of the research software ABAQUS, which is one of the largest universal finite element analysis programs, is becoming increasingly prevalent in the engineering industry. Because not only does it have a high speed, high accuracy, and low cost analysis of numerical calculation of finite element analysis software, but it also has a more user-friendly operator interface and visualisation results, particularly when it is utilised in the nonlinear analysis of precast concrete structures. Now, we make a conversation and analysis on the differences by comparing the test result data with the simulation result data of ABAQUS[1]. This can be a helpful reference for the further study of the finite element analysis on ABAQUS.

2 Methods

2.1 ABAQUS nonlinear analysis of reinforced concrete

Material nonlinearity, geometric nonlinearity, and nonlinear boundary conditions are the primary causes of nonlinearity in reinforced and precast concrete structures [2]. These three categories characterize the majority of the nonlinearity in these types of structures. When analysing the mechanical characteristics of steel and concrete, it is important to consider both the linear phase's elastic properties as well as the nonlinear stage's plastic properties. This is what is meant by the term "material nonlinearity." By defining the steel and concrete constitutive models in ABAQUS, we are able to accomplish the nonlinear properties of the material. We input the elastic modulus and Poisson's ratio of the two materials in the elastic stage. The definition of the plastic stage, on the other hand, is slightly different: reinforced simply enter the plastic stage of the stress-strain relationship; however, there are three models that can be selected in the plastic stage of concrete [3]: concrete smeared cracking, concrete damaged, and concrete damaged with spreading. In ABAQUS/Explicit, the plasticity and cracking model for concrete was created. The Plasticity model of concrete damage has a number of benefits, including its ability that it can be utilised in a variety of loads, including individual loads, repetitive loads, dynamic loads, and so on. Because it provides a satisfactory convergence, the Plasticity model of concrete damage is frequently utilised for the purpose of defining concrete plastic.

During the process of analysis, any changes made to the boundary conditions (including the interaction between the members) will result in the emergence of boundary nonlinear problems. Embedded technology allows ABAQUS to simulate the steel and concrete coming into frictional contact with one another. Through the use of integrated technology, reinforced unit is incorporated into concrete unit [4]. In the event that the magnitude of the displacement has an effect on the structural reaction, geometric nonlinearity will take place. The addition of the NLGEOM parameter, setting for a step is all that is required to determine whether or not ABAQUS will take into consideration geometric nonlinearity in that STEP option. However, the general nonlinear static analysis does not require the selection of NLGEOM parameters in order to avoid the laborious quantity of computation that would be required.

2.2 Analysis instance of simply supported beam made by precast concrete

2.2.1 The establishment of model

This beam is easily supported and measures 1100 millimeters in length, with a section that is 150 millimeters by 150 millimeters. The strength of the concrete is a C40 for precast and C20 for reinforced concrete. See Fig.1 for an explanation of the particulars of this scenario. The longitudinal reinforcement and stirrups employed HPB235 reinforced. In ABAQUS, the element C3D8R was used for concrete, and the element T3D2 was used for reinforced and precast concrete. In order to replicate the bonding relationship that exists between concrete and steel, we embedded reinforced concrete elements in concrete. In the event that there is a stress concentration in the beam loading surface and supports when we apply the load to the beam, we place a steel gasket in the acting location of the force and supports in order to increase the contact area and the stiffness: see Fig.2 for a model diagram of precast and reinforced beams.

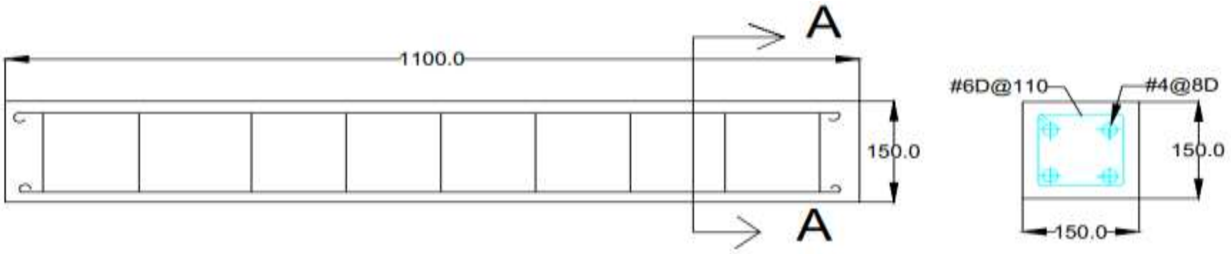
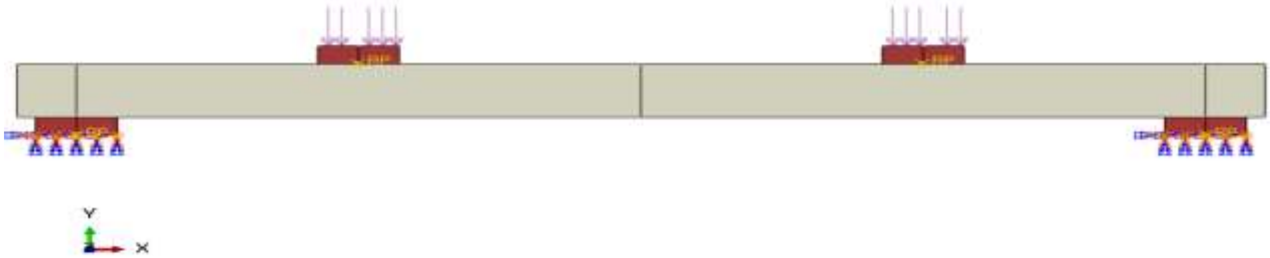


Fig.1 Geometry of precast and reinforced concrete

(a)



(b)

Fig.2 Model diagram of (a) reinforced concrete beam, (b) precast concrete beam

2.2.2 Calculation of the relevant parameters of reinforced concrete

The constitutive model of Reinforcement and precast uses bilinear model: the slope of the rising stage modulus of elasticity of steel, namely Es is 245GPa for reinforced concrete and 316GPa for precast concrete, and horizontal stage is from 0.001 to 0.006 of yield strain, with the yield stress of 245MPa for reinforced concrete and precast concrete correspondingly. Constitutive model for concrete axial compressive is based on the model formula suggested by E.Hognestad[5]:

$$\sigma=\begin{cases} f_c\left[2\frac{\varepsilon}{\varepsilon 0}-\left(\frac{\varepsilon}{\varepsilon 0}\right)^2\right] & \varepsilon \leqslant \varepsilon 0 \quad \cdots\cdots\cdots\cdots(1) \\ f_c\left[1-0.15\frac{\varepsilon-\varepsilon 0}{(\varepsilon u-\varepsilon 0)}\right] & \varepsilon 0 \leqslant \varepsilon \leqslant \varepsilon u \quad \cdots\cdots\cdots\cdots(2) \end{cases}$$

In which, uniaxial tensile strength of precast concrete is 14.2 MPa, yield strain is 0.002, and ultimate strain is 0.0038.

The constitutive model of concrete unidirectional tensile uses the formula in specification for design of precast concrete structure [6]:

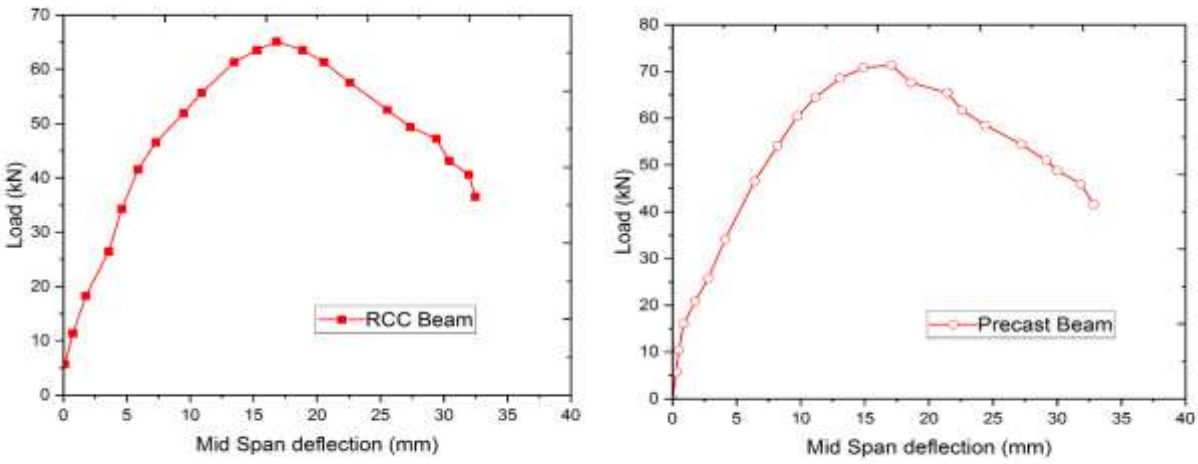
$$\sigma=\begin{cases} ft\left[1.2\frac{\varepsilon}{\varepsilon t}-0.2\left(\frac{\varepsilon}{\varepsilon t}\right)^{6}\right] & \varepsilon \leqslant \varepsilon t \quad \cdots\cdots(3) \\ ft\,\frac{\frac{\varepsilon}{\varepsilon t}}{(\alpha t)\left(\frac{\varepsilon}{\varepsilon t}-1\right)^{1.7}+\frac{\varepsilon}{\varepsilon t}} & \varepsilon \geqslant \varepsilon t \quad \cdots\cdots(4) \end{cases}$$

In which, uniaxial tensile strength of concrete is 1.51MPa, and concrete peak tensile strain is 73.64×10^{-6}.

3 Analysis results

3.1 Experiment

By design calculations, we determined the cross-sectional dimension, material parameters, amount of reinforcing steel and other data of the beam, carrying out simply supported at both ends and loads applying on two points, Test model shown in Figure 1. When testing, we apply stage loading in accordance with calculated capacity and the load on each point is the same. By controlling loading speed, we use a dial indicator and displacement gauge to measure the deflection of reinforced concrete beam, recording corresponding load, while observing the beam cracks and destruction till the beam completely destroyed. Drawn load-deflection curve of Reinforced and precast beams shown in Fig. 3.

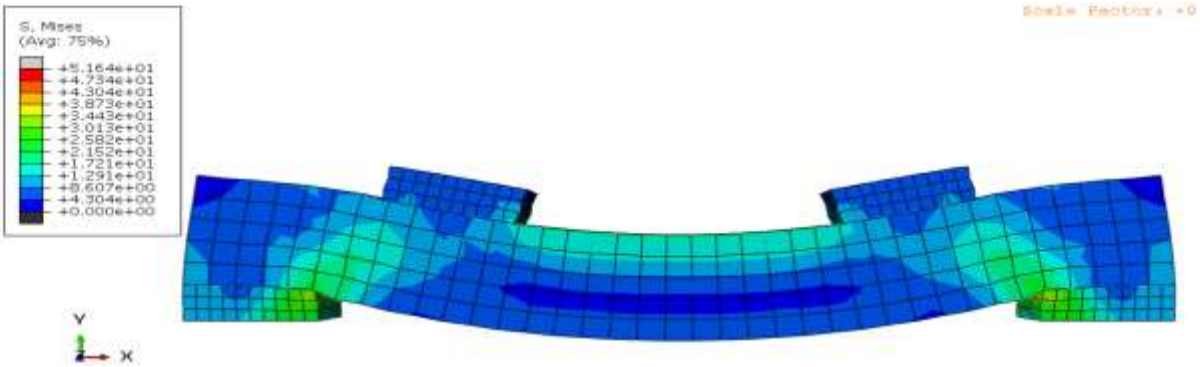


(a) (b)

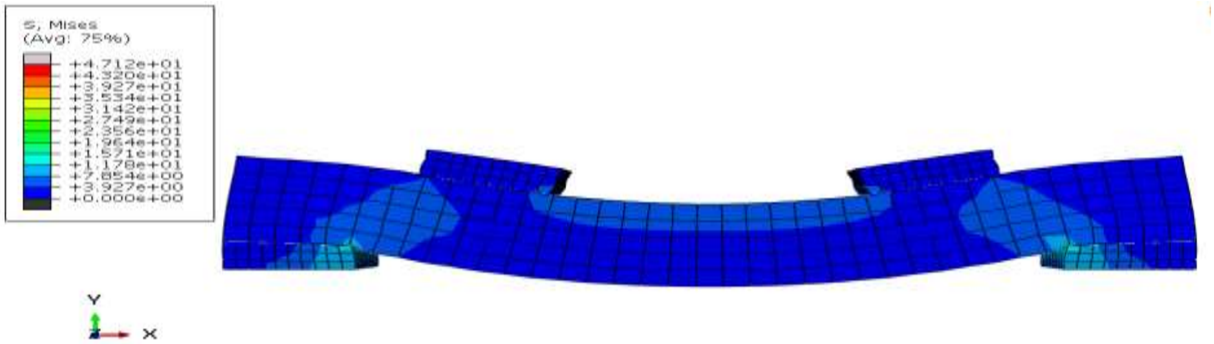
Fig. 3 load-deflection curve of (a) Rcc beam, (b) Precast beam

3.2 Simulation

We place a uniform load of 4.3 MPa on the gasket in a simply supported beam of reinforced concrete and 8.6 MPa on a beam composed of precast concrete with a force of 15 kN on the rcc beam and 28 kN on the precast beam respectively. The results of the measurement of the Mises stress of rcc and precast beams can be seen in Fig. 4. In a rcc beam, the gasket of support has a maximum tension of 51.64MPa, while a precast beam's gasket of support has a maximum stress of 47.12 Mpa. When force is applied to the gasket, a larger tension appears all around it. This stress increases from 5 MPa to 30 MPa in both beams. The stress distribution between two supports develops in a manner that is analogous to that of an arched truss force model of beam with web reinforcement. Both timbers are to have a stress arch. Under the impact of precast reinforcement, the Mises stress in the bottom of the beam is lower in comparison to the Mises stress in a normal rcc beam. This is due to the fact that the force acting on the beam shoulder is relatively low. When a uniform load of 4.3 MPa is applied to gasket beams made of reinforced and precast concrete, the displacement of nodes in the middle of the rcc beam is 9.310 mm, while the displacement of nodes in the centre of the precast beam is 4.580 mm (see Fig. 5). The time variations of this displacement are depicted in Fig. 6. As we can see, as the amount of time spent computing increased, the span deflection growth progressively picked up speed. After entering the plastic stage, the material property of reinforced concrete declines, and as a result, the acceleration of the deflection of the beam speeds up, thereby forming acceleration in the mid-span deflection curve. Since reinforced concrete in the elastic phase have high strength and strong rigidity, the amount of change in mid-span of deflection is small at the beginning. This demonstrates that the simulation is founded on an adequate theoretical foundation, making it an extremely trustworthy tool.

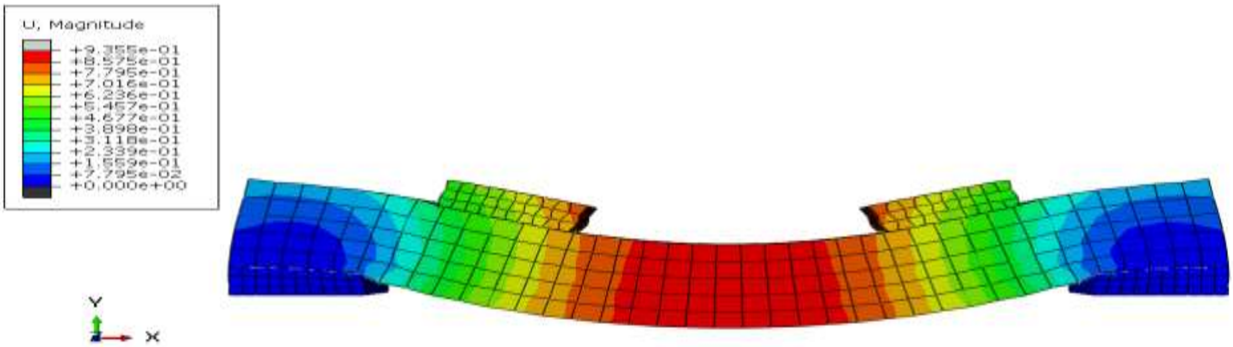


(a)

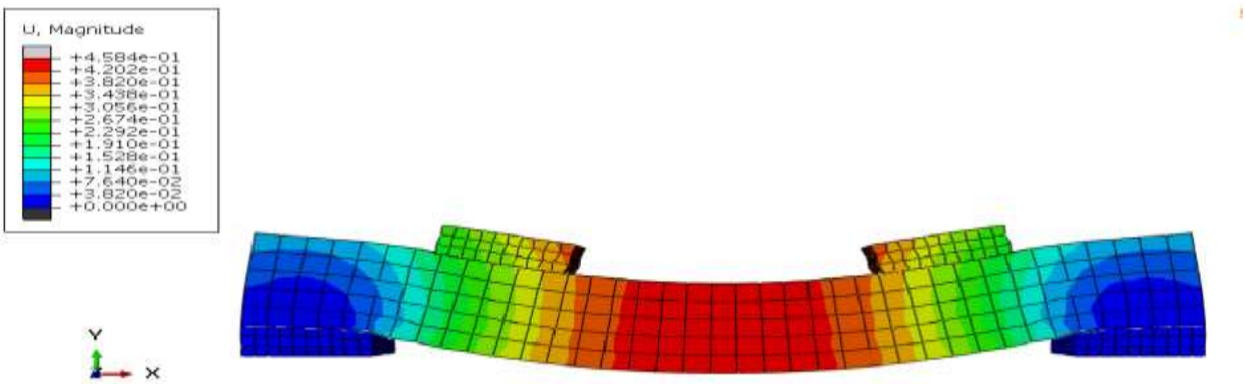


(b)

Fig. 4 Stresses developed in (a) monolithic beam, (b) precast beam

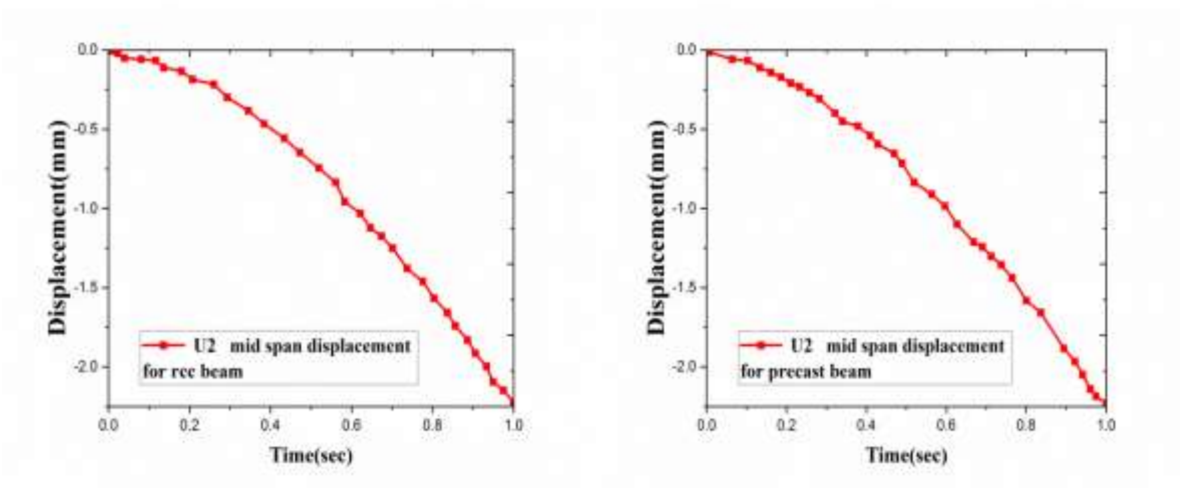


(a)



(b)

Fig. 5 Deflection in (a) monolithic beam, (b) precast beam

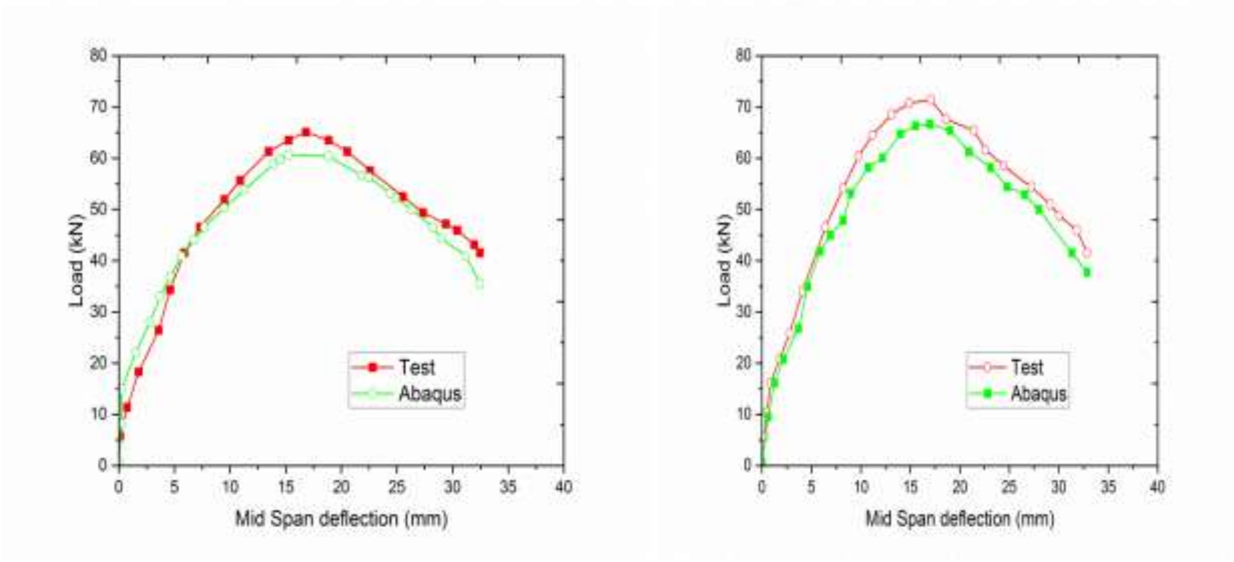


(a) (b)

Fig. 6 Mid span displacement-time curve for (a) monolithic beam, (b) precast beam

4. Comparison and Discussion

In order to carry out analysis and comparison between numerical simulation of Reinforced concrete beam and precast concrete beam conveniently, we use ABAQUS to apply the load step by step, calculating the results, getting the data of mid-span deflections and drawing the graphs. See Fig. 7.



(a) (b)

Fig. 7 mid span deflection comparison curve for (a) monolithic beam, (b) precast beam

Fig. 7 demonstrates that various patterns of mid-span deflection with load predicted by ABAQUS and those observed in the tests are identical. Before a weight of 35 kN in rcc and 43 kN in precast beam, the beams are in the elastic stage, exhibiting high stiffness and strength, with load values varying linearly with the amount of deflection. After that, it entered the plastic stage, where the process of adding deflection value intensified. When the load achieved its capacity of 65 kN, the value of the mid-span deflection in ABAQUS was 15.251mm, whereas the value measured in an RCC beam test was 14.75mm. In a scenario that is similar when the weight on a precast beam reaches its capacity of 73 kN, the value of the beam's mid-span deflection is 18.34mm in ABAQUS and 15.24mm. It is clear that ABAQUS can generally be consistent with the results of the actual tests, but it is also clear that there are some discrepancies between the values determined by the analysis and the results of the tests. Possible explanations include the following:

(1) The contact form between cells in a finite element simulation is supposed to be uniform, isotropic, and the same; however, the composition of real concrete, which includes cement, sand, gravel, and other materials, is very complex. It is difficult to simply substitute their convoluted interaction with a single form.

(2) Finite Element Analysis processes the bond between concrete and steel using embedded technology, which significantly simplifies modelling. However, this bond cannot be accomplished with reinforced slip simulation and the increased load of reinforced concrete. Results may be easily distorted as a consequence of this.

(3) The convergence of the finite element analysis, the reasonableness of the simulation parameter values, the form and number of finite element divisions, the casting quality, and the loading conditions of the test beam are additional variables that influence the analysis's outcomes.

In simple terms, the finite element analysis simulation test of reinforced concrete and precast concrete has a high degree of similarity with the actual situation. Regardless of the fact that there are some differences, the finite element technique for nonlinear analysis of beam test is achievable.

5. Summary

In this article, the authors discussed the factors that contribute to the disparities in the findings of numerical analyses performed on reinforced concrete and precast concrete. In this paper, we imitated the test of a beam by using the software ABAQUS. We did this by adopting a model of concrete damaging and the technology of embedded between steel bar and concrete. As a result, we obtained findings that were comparable to those obtained from actual tests of reinforced concrete beams and precast concrete beams. All of these things demonstrate that the finite element software ABAQUS is efficient enough to compare the components of a reinforced concrete beam and a precast concrete beam.

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 school of civil engineering, Vellore institute of technology, Vellore. These supports are gratefully acknowledged.

Notation

The following symbols are used in this paper:

FEA = finite element analysis

E= modulus of elasticity of concrete

σ =yield stress

$\varepsilon_=$ Yield strain

$\varepsilon 0_=$ Yield strain in plastic zone

ft= Ultimate strength compression zone

fc= Ultimate strength tension zone

C40=Concrete compressive strength of 40 MPa

C20=Concrete compressive strength of 20 MPa

ε_t =tensile plastic strains

ε_u = compressive plastic strains

α_t= tensile damage parameter

C3D8R= 8-node linear brick, reduced integration with hourglass control

T3D2= two-node, 3-dimensional truss element

References

[1] L Jing-Song, Liu Hong-Jun. ABAQUS Finite Element Analysis of Reinforced Concrete [J]. *Equipment Manufacturing Technology*, 2009,(6):69-70,107.

[2] Lv Xi-Lin, Jin Guo-Fang, Wu Xiao-Han. Reinforced Concrete Structure Nonlinear Theory and Applications [M]. *Shanghai: Tongji University Press*, 1997: 55-67.

[3] Zhuang Zhuo, Zhang Fan, Cen Song. ABAQUS Nonlinear Finite Element Analysis and Examples[M]. *Beijing: Science Press*, 2005:123-139.

[4] ABAQUS Analysis User’s Manual， *ABAQUS Inc*, 2006.

[5] Southeast University etc. Concrete Structure Design Theory[M]. *Beijing: China Building Industry Press*, 2008.

[6] Design Principle of Concrete Structures, GB50010 – 2010.

[7] JIANG Jian-jing, LU Xin-zheng, YE Lie-ping. Finite Element Analysis of Concrete *Structures* [M]. Beijing: Tsinghua University Press, 2005.

[8] ZHANG Guo-li, SU Jun. Based on ABAQUS Nonlinear Analysis of Reinforced Concrete [J]. *Science Technology And Engineering*, 2008,8(20):5620-5624.

[9] Hawileh R A, Rahman A, and Tabatabai H 2010 Nonlinear finite element analysis and modeling of a precast hybrid beam-column connection subjected to cyclic loads. *Applied Mathematical Modelling*, 34(9): 2562–2583. https://doi.org/10.1016/j.apm.2009.11.020

[10] Kulkarni S A, Li B, and Yip W K 2008 Finite element analysis of precast hybrid-steel concrete connections under cyclic loading. *Journal of Constructional Steel Research*, 64(2): 190–201. https://doi.org/10.1016/j.jcsr.2007.05.002

[11] Kaya M, and Arslan A S 2009 Analytical modeling of post-tensioned precast beam-to-column connections. *Materials and Design*, 30(9): 3802–3811. https://doi.org/10.1016/j.matdes.2009.01.033

[12] ABAQUS 6 14 2014 ABAQUS 6.14 Analysis User's Guide, Volume IV: Elements. *ABAQUS 6.14 Analysis User's Guide*, *IV*, 1–1128. http://130.149.89.49:2080/v6.14/books/usb/default.htm

[13] Feng D C, Wu G, and Lu Y 2018 Finite element modelling approach for precast reinforced concrete beam-to-column connections under cyclic loading. *Engineering Structures*, 174(November): 49–66. https://doi.org/10.1016/j.engstruct.2018.07.055

[16] Genikomsou A S, and Polak M A 2015 Finite element analysis of punching shear of concrete slabs using damaged plasticity model in ABAQUS. *Engineering Structures*, 98: 38–48. https://doi.org/https://doi.org/10.1016/j.engstruct.2015.04.0