OPEN ACCESS
Saudi Journal of Civil Engineering
Abbreviated Key Title: Saudi J Civ Eng
ISSN 2523-2657 (Print) |ISSN 2523-2231 (Online)
Scholars Middle East Publishers, Dubai, United Arab Emirates
Journal homepage: https://saudijournals.com/journal/sjce/home

Original Research Article

Study of Crack Pattern in RCC Beam Using ABAQUS

Ashutosh Kumar Yadav[1*], Ashish Poudel[1]

[1]Department of Civil Engineering, Gayatri Vidya Parishad College of Engineering (A), Visakhapatnam - 530048, India

DOI: 10.36348/sjce.2021.v05i05.003 | **Received:** 16.05.2021 | **Accepted:** 20.06.2021 | **Published:** 23.06.2021

***Corresponding author:** Ashutosh Kumar Yadav

Abstract

Crack formation and propagation in reinforced concrete members is complex. It involves various mechanisms and parameters related to the interaction between concrete and reinforcement, the geometry of structural members, type of loading, and support conditions. These factors can be controlled and adjusted to some extent only. On the other hand, these deterministic factors are all superimposed by a certain level of randomness owing to the fluctuations of material parameters and the heterogeneous composition of concrete. Nevertheless, the process of cracking and the resulting crack systems have been widely studied during the years, striving to create physically transparent models that could predict the structural behavior with greater accuracy. Considering the importance of flexure and shear failure influences on strength of concrete, percentage of steel and stirrups of RC beam, a finite element (FE) modeling that focuses on crack initiation and pattern is presented in this study. Nonlinear three dimensions finite element analysis (FEA) of RC beam capable of appropriately modeling the concrete stress-strain behavior, tensile cracking, and compressive damage of concrete and indirect modeling of the steel-concrete bond is performed using ABAQUS to investigate the ultimate load, beam displacements, and cracking pattern. The concrete damage plasticity is applied to the numerical model as a distributed plasticity over the whole geometry. The objective of this study is to predict failure and crack development in the concrete model using ABAQUS for static load in M30 grade beam with different reinforcement configurations and results have been compared with the experimented sample.

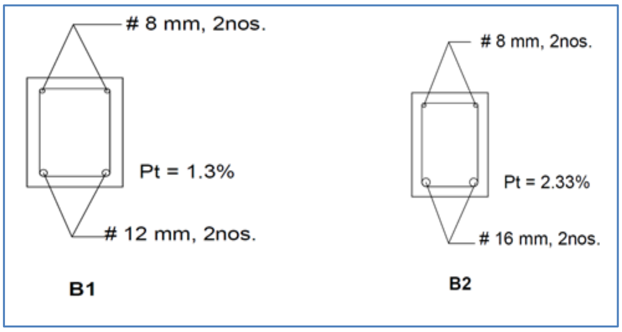
Keywords: Beam, Finite Element Analysis, concrete damaged plasticity, crack pattern.

I. INTRODUCTION

Reinforced concrete (RC) is one of the most important building materials and is widely used in many types of engineering structures. The economy, efficiency, strength, and stiffness of reinforced concrete make it an attractive material for a wide range of structural applications. Understanding the response of the structural components during loading is crucial to the development of an overall efficient and safe structure. Different methods have been utilized to study the response of structural components. The results are compared with the theoretical calculations that estimate deflections at the serviceability load of the beams. Finite element analysis can also be used to model the behavior numerically to confirm these calculations, as well as to provide a valuable supplement to laboratory investigations, particularly in parametric studies. With the continuous development of finite element theory and computer technology, the development of finite element analysis software is maturing. ABAQUS, as one of the largest universal finite element analysis software, is increasingly commonly used in research works and engineering. Because not only does it have high speed, high accuracy, and low-cost analysis of the numerical calculation of finite element analysis software, but also has a more user-friendly operator interface and visualization results, especially when it is used in the nonlinear analysis of reinforced concrete structure [1]. The numerical results from the FEA from ABAQUS are compared with the experimental results which showed good agreement between the results. Laboratory tests were carried out for plain, under, balanced, over reinforced sections. The failure mechanism of a reinforced concrete RC beam was modeled quite well using FEA and the failure load predicted was very close to the failure load measured during experimental testing. [2]. Simply supported RC beam analysis with the plasticity model of concrete damage in ABAQUS when compared with the experimental result revealed that FEM analysis result was found similar to numerical analysis so can be a reference for the further study of the nonlinear analysis of reinforced concrete [3]. A model was developed by taking into account the effect of compressive strain localization. Approaches to the application of fracture mechanics to the concrete are discussed, with an emphasis on the models based on strain softening and strain localization, particularly the fictitious crack model [4]. A fracture mechanics model for bending of RCC beams where concrete is assumed to be a linear elastic material and steel is considered to be elastic-plastic was proposed. The effect of reinforced steel bars is simulated by a closing force whose magnitude is determined by a compatibility condition [5]. A fracture mechanics approach to predict cracking of RC members subjected to tension by balancing the rates of change of the strain energy, the debonding energy, and the sliding energy was developed [6].

II. METHODS

2.1 Model establishment



The simply supported beam is 1500 mm long with an effective length of 1300 mm and a cross-section of 100 mm width and 200 mm depth with 20 mm clear cover is adopted. Support length of 150 mm was taken from both ends. Two beam specimen B1 & B2 with the same section, length, confinement detailing, and spacing but the different percentages of steel in B1 and B2 were taken for this analysis: see Figure 1. Concrete of strength 30 N/mm^2, density $2.5x10^{-5}$ N/mm^3, and Poisson's ratio 0.15 was used for this analysis. Longitudinal reinforcement with a yield stress of 500 N/mm^2 (Fe500) and confinement bars with a yield stress of 415 N/mm^2 (Fe415) was used. The density of steel $7.5x10^{-5}$ N/mm^3, young's modulus 200000 N/mm^{2}, and Poisson's ratio 0.3 are used. In ABAQUS, the concrete adopted the C3D8R element, and the reinforced used the T3D2 element. We embedded reinforced concrete elements to simulate the bonding relationship between the reinforced and concrete. In case of stress concentration in beam loading surface and supports when we apply the load on the beam, we set steel gasket in the acting position of the force to increase the contact area and stiffness as shown in Fig 2 for model diagram.

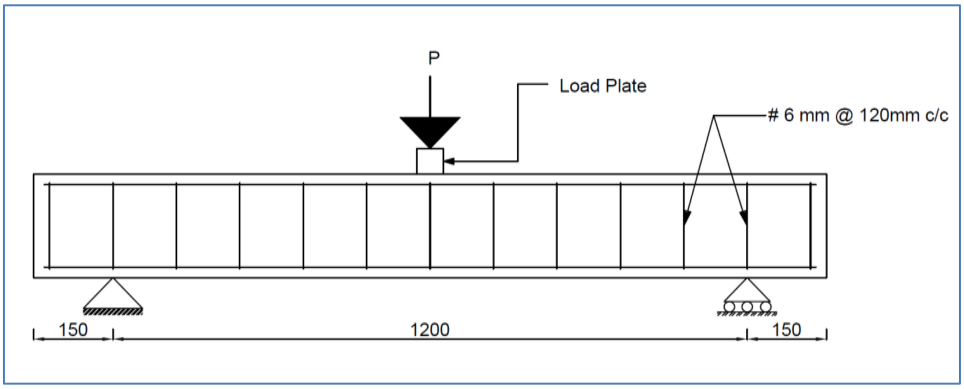


Fig-1: Beam gometry

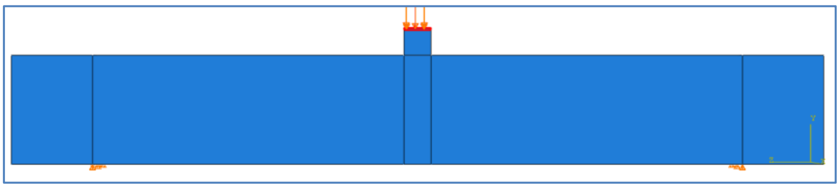


Fig-2: load and boundary condition

2.2 Concrete damaged plasticity

The Concrete Damage Plasticity (CDP) is the most comprehensive continuum model that was used in the reinforced concrete beam simulation to define concrete behavior in this analysis. The CDP model applies to concrete that is subjected to monotonic loading for different types of structures (such as beams, trusses, shells, and solids) and it is developed based on two concrete failure mechanisms. a) Compressive crushing b) Tensile cracking.

For concrete subjected to uniaxial loading, the stress-strain response of concrete in tension exhibits a linear elastic relationship until the failure stress is achieved, and beyond that point, the concrete follows the softening stress-strain behavior. When the concrete is unloaded at any point from within the strain-softening portion of the curve, the unloading response is weakened and the elastic stiffness of concrete is damaged. This deterioration of the stiffness is defined by the damage parameter in tension (d_t), which can range from zero representing the undamaged condition of the specimen to one which signifies that the material has lost its total strength.

Before defining the concrete damaged value in Abaqus the parameters required defining the yield surface consists of four constitutive parameters. The Poisson's ratio controls the volume changes of concrete for stresses below the critical value which is the onset of inelastic behavior. Once the critical stress value is reached concrete exhibits an increase in plastic volume under pressure (Chen, 1982). This behavior is taken into account by defining a parameter called the angle of dilation. In the CDP model, 'ψ' is the dilation angle measured in the p-q plane at high confining pressure, and in this study, it is determined with sensitivity analysis. ϵ is an eccentricity of the plastic potential surface with a default value of 0.1. The ratio of initial biaxial compressive yield stress to initial uniaxial compressive yield stress, 'fb_0/fc_0', with a default value of 1.16. Finally 'K_c' is the ratio of the second stress invariant on the tensile meridian to the compressive meridian at initial yield with a default value of 2/3 (Abaqus User Manual, 2014). The parameter 'K_c' should be defined based on the full triaxial tests of concrete, moreover, a biaxial laboratory test is necessary to define the value of 'fb_0/fc_0'. This paper does not discuss the identification procedure for parameters 'ϵ', 'fb_0/fc_0', and 'K_c' because tests that are going to be verified in this study do not have such information. Thus, default values are accepted in this study.

Table-1: Plasticity damaged parameter

ψ	E	fb0/fc0	Kc	μ
35	0.1	1.16	0.667	0

2.3 Stiffness recovery

Default values for stiffness recovery factors are assumed in the present study. Therefore, the compressive stiffness recovery factor, w_c = 1 is used assuming that compressive stiffness is fully recovered upon crack closure as the load changes from tension to compression, and the tensile stiffness recovery factor, w_t = 0 is used assuming that tensile stiffness is not recovered as the load changes from compression to tension once crushing of concrete is initiated. The rebars and concrete cracking behavior were presumed to behave independently.

2.4 The finite element mesh

To get accuracy in results, all the elements of the FE model were assigned the same mesh size so that every two different materials can share the same node among them. After assembling and assigning the properties, an input file is created which is then imported to create a mesh. The elements used for the study are C3D8R (8-node linear brick) and T3D2 (2-node linear 3-D truss. The concrete beam is modelled in 3-D assigned with C3C8R elements and reinforcement in the longitudinal direction, and the shear reinforcement bar (Stirrups) is modelled in 2-D was assigned with the T3D2 element. Meshing was adopted for all of the elements used in the models. Also, the aspect ratios of solid elements were kept as close to one as practicable, as high aspect ratio elements would affect the accuracy of the analysis. Mesh convergence studies were conducted to determine the best balance between accuracy and computational cost. Uniform element sizes of 10 mm were adopted for all specimens Shown in Fig: 5. Frictional contact of the steel and concrete in ABAQUS is achieved by embedded technology. Reinforced unit is embedded in concrete unit by embedded technology [9].

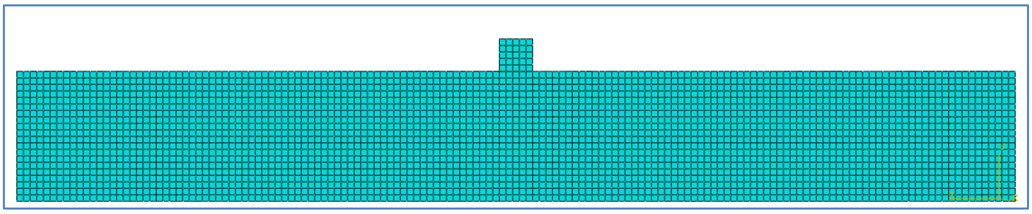


Fig-5: Meshing size of 10 mm

2.5. Simulation

According to Lubliner et al. [10], cracks occurred primarily in the concrete equivalent plastic strain is greater than zero, namely: ε_t ~p_l >0, and the maximum principal plastic strain is a positive point. The crack development direction, width, and distribution status can be observed following the minimum plastic strain in ABAQUS. When the value of the minimum principal strain reached the limits of concrete compressive strain (ε=0.0033), indicating that the concrete crushing at the location. Initial crack is vertical cracks that appeared mainly in the center of the beam. With the increase of load, crack continues to grow and various cracks started to grow in the increased range at the same time when the yield point is reached, the occurs at the beam bottom. When the load reaches the limit state, the concrete compression zone reaches the maximum compressive stress, concrete failure begins.

III. SIMULATION RESULT

3.1. Load-displacement behavior

Load and deflection are directly proportional as deflection increases with the increase of load acting on the beam. To obtain the ultimate load capacity of the beam, the load was gradually increased to the fracture point. A load versus deflection curve was plotted to study the ultimate load-carrying capacity of the beam. The ultimate load-carrying capacity of B1 was found 69 kN and B2 was found 121 kN with 5 mm and 11 mm displacement respectively. The curve shows an almost proportional variation of displacement with an increase of load till the ultimate load which can be represented by the peak of the curve, after which the curve descends showing the plastic properties of concrete Figure: 6. The visualization pattern of displacement of beam created by Abaqus is presented in fig: 7.

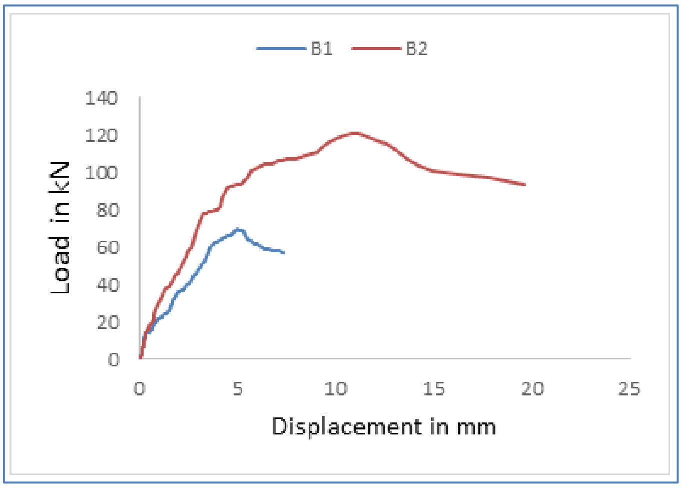
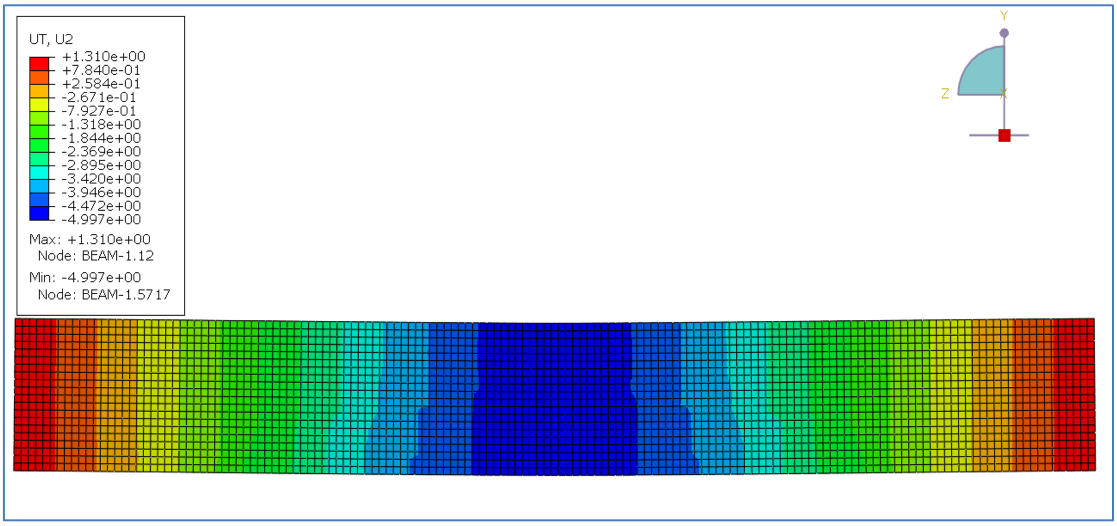
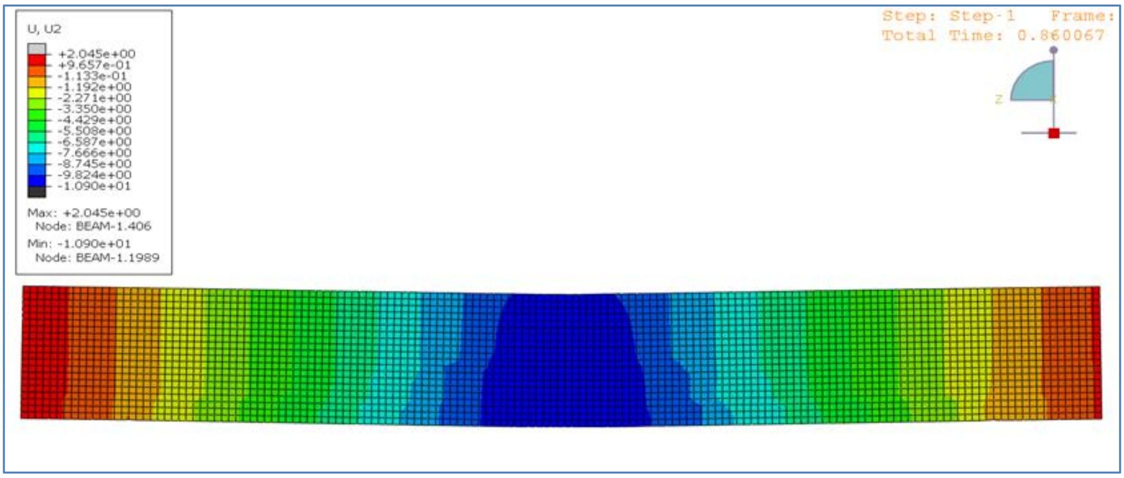


Fig-6: load vs displacement relation



a) Displacement in B1



b) Displacement in B2

Fig-7 (a & b): Displacement at ultimate load

Maximum permissible displacement at serviceability load is span/250 equal to 6 mm. B1 and B2 deflects 2.8 mm and 4 mm respectively at their serviceability load which falls under the limit illustrated in table 2.

Table-2: Serviceability load and displacement

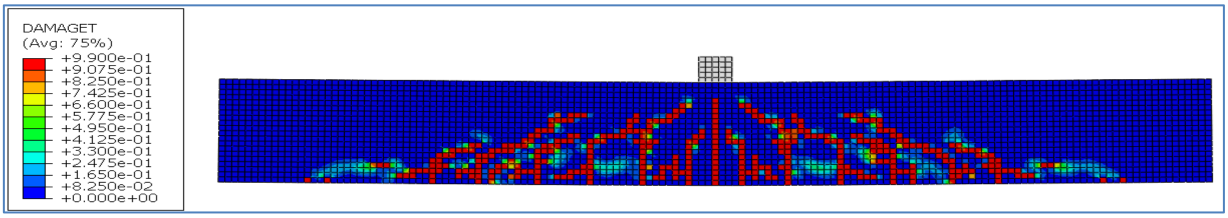
Specimen	Design load (kN)	Ultimate load (kN)	Serviceability load (kN)	Displacement at serviceability load (mm)
B1	47	69	46	2.8
B2	82	121	81	4

3.2 First crack

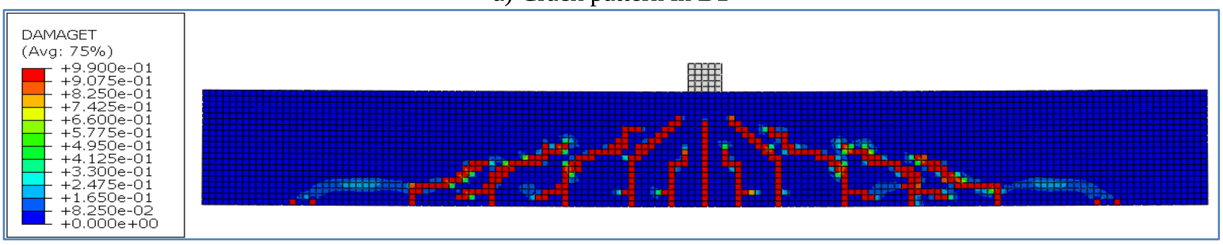
The first crack in B1 was witnessed at the load level of 16 kN with a displacement of 0.3 mm when the value of the cracking strain reached the limits of 0.002 and the first crack in B2 was witnessed at the load level of 23 kN with a displacement of 0.2 mm when concrete compressive strain exceeds the value of 0.00352. As the load level was increased, further cracks were developed in other portions of the beam in the specimens.

3.3 Crack pattern at ultimate load

In both beams, first cracks were initiated at different load levels in the mid-span, and they propagated in the vertical direction. As the load increased, more flexural cracks formed within the mid-span and toward the support regions, and the existing vertical cracks became insignificantly wider and deeper. With further increase in loading, vertical cracks situated close to the supports started to change their orientation and become the inclined cracks. The distribution of crack patterns varied according to the reinforcement ratio variation illustrated in the Figure: 8.



a) Crack pattern in B1

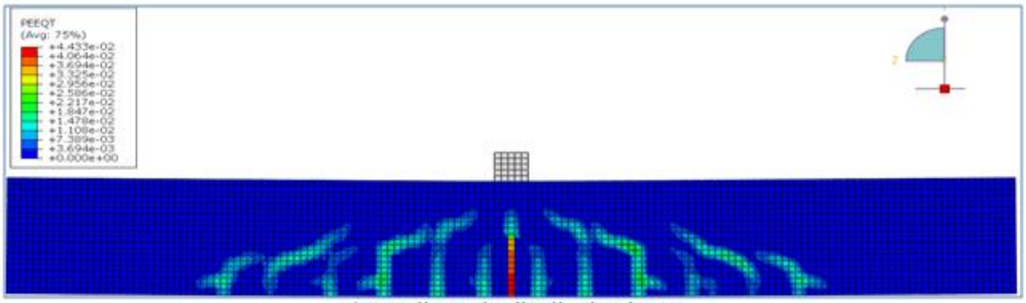


b) Crack pattern in B2

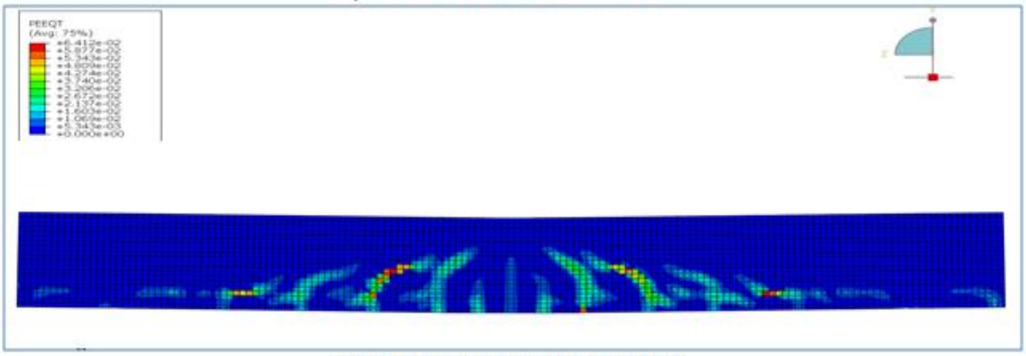
Fig-8: (a & b) crack pattern at ultimate load

3.4 Tensile strain in concrete at ultimate load

In the below fig: 9 PEEQT refers to tensile equivalent plastic strain also known as cracking strain tells about the intensity of crack opening concerning another crack at a particular applied load.



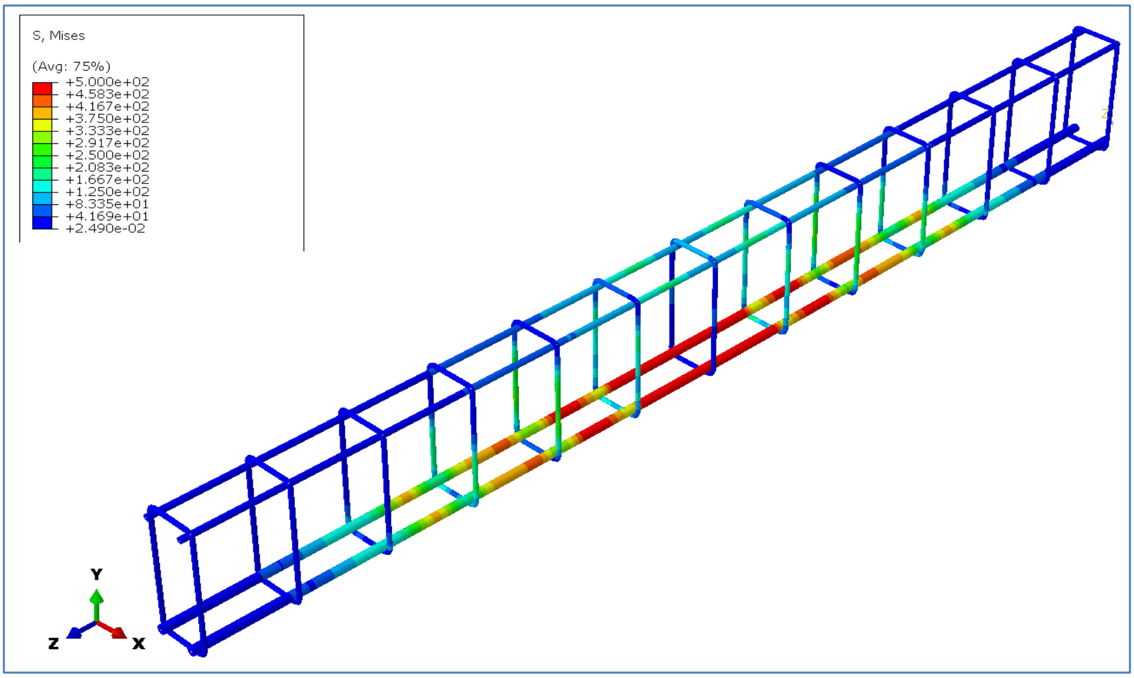
a) Tensile strain distribution in B1



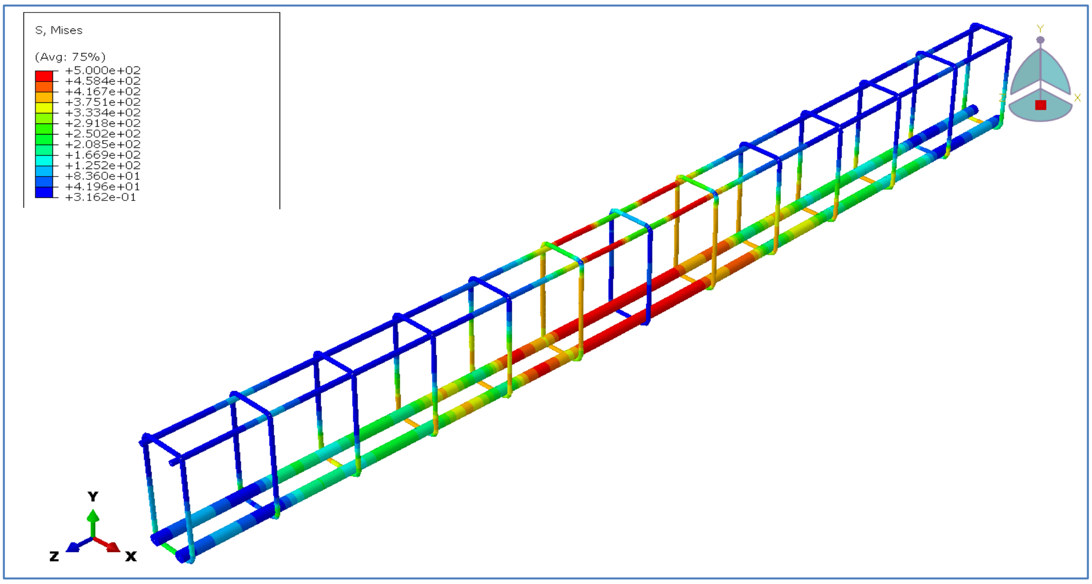
b) Tensile strain distribution in B2

Fig-9: (a & b) tensile strain pattern distribution in concrete at ultimate load

3.5 Stress in reinforcement in B1 and B2 at ultimate load



a) Stress distribution in reinforcement B1

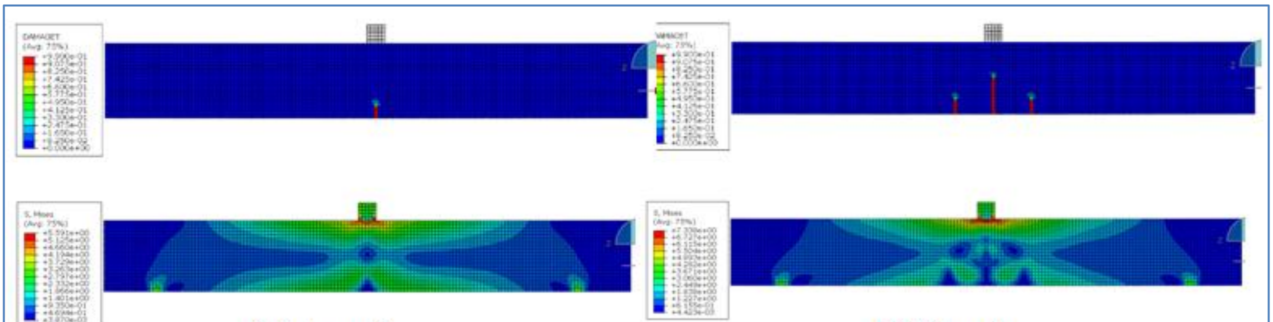


b) Stress distribution in reinforcement B2

Fig-10 (a & b) stress distribution in reinforcement at ultimate load

3.6 Stress Softening

In figure: 11 it was observed that at the location of the crack, stress is softened which causes initiation of crack, and stress concentration was observed on the first crack tip in a result the first crack stops there and promotes other cracks to grow at other places. Figure: 12 experiments beam and Figure: 13 is simulated model of the beam, comparing the location of the flexural crack is similar. But crack path somewhat differs from than analyzed one because in practical, fracture energy depends on the size and crushing strength of aggregate that comes into account for crack path and propagation which lacks in the simulation.



a) first crack   b) 2nd crack

Fig-11: formation of crack due to stress softening

Fig-12: Experimented RCC beam

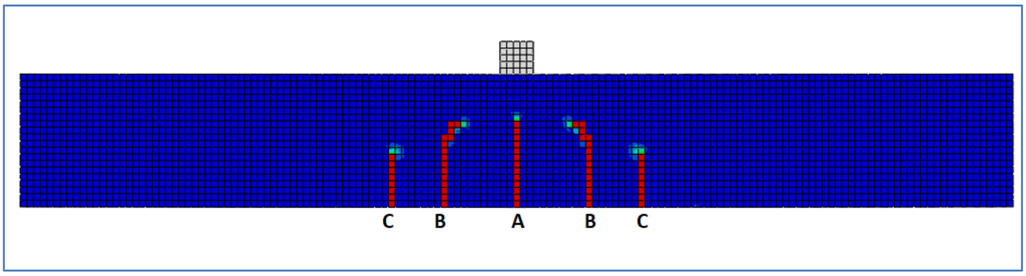


Fig-13: Major flexural cracks

IV. DISCUSSION

Nonlinear finite element analysis of two RCC beam B1 and B2 was performed to capture the first crack load, crack pattern at various load, serviceability load, ultimate load, displacement, stress, and strain pattern in concrete and reinforcement. Concrete damage plasticity material model is applied to the numerical procedure as a distributed plasticity over the whole geometry of the specimens to appropriately simulate material nonlinearity. B1 and B2 of the same cross-section and length with varying percentages of tension reinforcement are studied. Percentage reinforcement P_t of 1.3 % and 2.33 % are taken.

V. CONCLUSION

1. The ultimate load capacity was found to be 69 kN and 121 kN for B1 and B2 respectively.
2. An increase of 79% of P_t (Percentage of tensile reinforcement) leads to a rise in ultimate load-carrying capacity by 75% or 1.75 times in B2.
3. The serviceability load was found lesser than the design load.
4. Both beam specimens were under permissible displacement limit at serviceability load.
5. The first crack was noticed in 16 kN load for B1 while 23 kN load was required for B2 to initiate its first crack.
6. Before the first crack, the stress in B1 was found to be 3.028 N/mm^2 but when crack started initiating stress at crack was declined to 2.256 N/mm^2 at the crack location, similarly before first crack stress in B2 was found to be 2.78 but N/mm^2 when the crack starts initiating stress decline to 0.017 N/mm^2 at crack location. The decline of stress after occurring of crack is stress softening which promotes the initiation and propagation of the cracks.
7. Stress concentration occurs at the tip of the crack which forms the cloud of plastic strain above the tip which makes the crack tip blunt, in a result crack stops there and starts growing at other places.

VI. REFERENCES

1. LIU, J. S., & LIU, H. J. (2009). Abaqus finite element analysis of reinforced concrete. Equipment manufacturing technology, 6, 30.
2. Tejaswini, T., & Raju, M. V. R. (2015). Analysis of RCC Beams using ABAQUS. International journal of Innovations in Engineering and Technology (IJIET), 5(1).
3. Buckhouse, E. R. (1997). External flexural reinforcement of existing reinforced concrete beams using bolted steel channels.
4. 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. Atlantis Press.
5. Hillerborg, A. (1988). Applications to Fracture Mechanics to Concrete. Report TYBM-3030, Lund, Sweden, 1-28.
6. Bosco, C., Carpinteri, A. (1992). Fracture mechanics evaluation of minimum reinforcement in concrete structures. Application of fracture mechanics to reinforced concrete, Proc. Int. Workshop, Turin, Italy, 1990, Ed. Elsevier, London, 347–377.
7. Shah, S. P., Swartz, S. E., Ouyang, C. (1995). Fracture mechanics of concrete. John Wiley & Sons, New York.
8. LV, Xi-lin., JIN, Guo-fang., WU Xiao-han. (1997). Reinforced Concrete Structure Nonlinear Theory and Applications [M]. Shanghai: Tongji University Press, 55-67.
9. Zhuang, Z., Zhang, F., & Cen, S. (2005). ABAQUS nonlinear finite element analysis and examples.
10. ABAQUS analysis users's manual 2014
11. Lubliner, J., Oliver, J., Oller, S., & Oñate, E. (1989). A plastic-damage model for concrete. International Journal of solids and structures, 25(3), 299-326.

