Al-Azhar University Civil Engineering Research Magazine (CERM)
Vol. (41) No. (1) January, 2019

Three-Dimensional Non-Linear Analysis of Two-Span Reinforced Concrete Beam _ Using the Finite Element Code ABAQUS

O.A. Hodhod [1], A.M. Sanad [2], M.M. El-Attar [1], H.A. Hassan [3]

[1] Prof. of Properties and Strength of Materials, Str. Eng. Dept., Faculty of Engineering, Cairo University, Egypt
[2] Prof. of Structural Engineering, Construction and Building Dept., College of Engineering, Arab Academy for Science & Technology, Cairo, Egypt
[3] Lecturer Assistant, Construction and Building Dept., College of Engineering, Arab Academy for Science & Technology, Cairo, Egypt.

ملخص البحث

تقدم هذه الدراسة محاولة ناجحة لنمذجة كمرة خرسانية مسلحة ذات بحرين باستخدام كود العناصر المحددة ABAQUS. تم إجراء تحليل غير خطى باستخدام خيار اللدونة المتضررة للخرسانة والموجود فى برنامج ABAQUS 6.14. تم اقتراح نموذج معدل لتقوية الخرسانة فى الشد وتم استخدامه فى النموذج. تم التحقق من صحة نتائج نموذج العناصر المحددة باستخدام نتائج كمرة تجريبية تم اختبارها في اختبار الانحناء. يتم عرض النتائج من خلال منحنيات التحميل مع الهبوط، وأنماط التشرخ، وخطوط الاجهادات في كل من مناطق الضغط والشد. أظهرت النتائج اتفاقا جيدا بين نموذج العناصر المحددة والكمرة التجريبية.

Abstract

This paper presents a successful attempt for modeling a two-span reinforced concrete beam using the finite element code ABAQUS. A non-linear analysis is performed using the concrete damaged plasticity option in ABAQUS 6.14. A modified tension stiffening model is proposed and used in the model. The results of the finite model are validated using results of an experimental beam tested in flexural test. The results are presented in terms of load-deflection curves, cracking patterns, and stresses contours in both compression and tensile zones. The results showed good agreement between the finite and the experimental beam.

Keywords

ABAQUS, Damage Plasticity, Non-Linear, Reinforced concrete, Tension Stiffening.

1. Introduction

Concrete is the most popular construction material around the world. It is preferred in construction projects for its durability, versatility, mechanical and physical properties. However, its non-linear behavior is very complex. Finite element analysis of concrete as a non-linear material can be a useful way for modelling concrete. Many finite element packages are commercially available, such as ADINA, ABAQUS, NASTARAN, ANSYS, etc.

Researchers found that this complexity came from the non-linear stress-strain relation of the concrete under multi-axial stress conditions, strain softening and anisotropic stiffness reduction, progressive cracking caused by tensile stresses and strains, and bond between concrete and embedded rebars [1]. Therefore, an appropriate analysis of reinforced concrete, using a finite element software, depends totally on the input data, especially the material properties used [2]. Standard uniaxial compressive strength tests on concrete samples showed highly non-linear stress-strain relationship. As an attempt to model the non-linear behavior of concrete, a numerical expression has been developed by Hognestad [3],[4]. The model simulates the behavior of concrete as parabola in the ascending part and the descending part as a straight line as shown in equations 1,2.

$$For\ 0 < \varepsilon < \varepsilon_0', \qquad \frac{\sigma}{\sigma_{cu}} = 2\frac{\varepsilon}{\varepsilon_0'}\left(1 - \frac{\varepsilon}{2\varepsilon_0'}\right) \tag{1}$$

$$For\ \varepsilon_0' < \varepsilon < \varepsilon_{cu}, \qquad \frac{\sigma}{\sigma_{cu}} = 1 - 0.15\left(\frac{\varepsilon - \varepsilon_0'}{\varepsilon_{cu} - \varepsilon_0'}\right) \tag{2}$$

In ABAQUS, concrete as a non-linear material can be modeled using the concrete damaged plasticity model. This model is modified from the Drucker-Prager criterion [5]. To represent the inelastic behavior of concrete in association with its isotropic damaged elasticity, this model takes into consideration the isotropic compressive and tensile plasticity of concrete. Also, the cracking in tension or crushing in compression are considered in this model. George et al. [6] used the ABAQUS package to simulate the damage induced plasticity in concrete. Three-point bending test has been carried out on beams of various sizes. Concrete damaged plasticity model has been used to carry out the analysis.

According to Alrazi et al. [7], the use of implicit procedure to solve problems involving cracking and damage is not always efficient as convergence difficulties are often encountered, whereas a dynamic explicit analysis can be more efficient to model complex nonlinear material behavior involving damage and large deformations. To ensure a quasi-static solution is obtained with the dynamic explicit procedure, the displacement must be applied smoothly and slowly to eliminate any significant change in the acceleration from one increment to the other [7].

2. Materials and Experimental Test Set-Up

2.1 Materials

Two-span reinforced concrete beam with dimensions 100 mm width, 250 mm height, and 3000 mm total length was designed and casted. The beam's reinforcement detailing is shown in Fig. 1. The beam is casted using Ordinary Portland Cement CEM I 42.5R. The target strength is 40 MPa after 28 days. The maximum size of the coarse aggregate is limited to 20 mm to agree with the beam dimensions and reinforcement distribution.

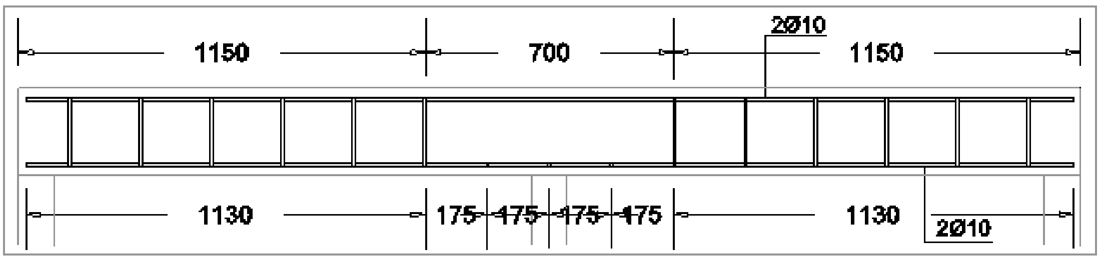

Figure 1: Beam's reinforcement detailing, dimensions in mm

2.2 Experimental test set-up

Steel loading frame was designed and fabricated for testing the RC beam in a two-span flexural test. Two edge supports, and one middle support are provided for the beam to be carried and two reaction beams are located at the top of the frame and at the middle of each span. Figure 2 shows a schematic drawing for a bean loaded in the loading frame.

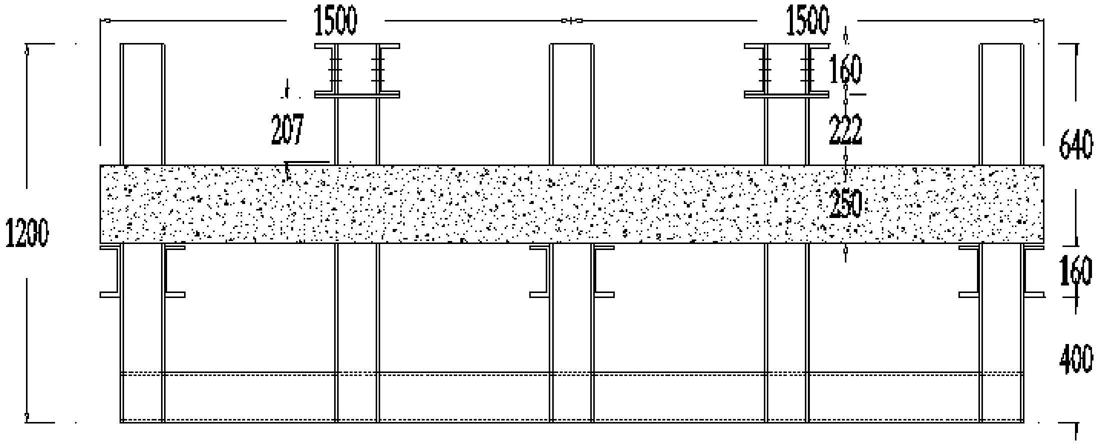

Figure 2: Schematic drawing for a beam loaded in the loading frame, dimensions in mm

3. Finite Element Model

The experimental RC beam is modeled using the finite element software package ABAQUS 6.14. The geometry and the reinforcement of the modeled beam are typically the same as the experimental beam. The geometry, mesh, boundary conditions, and supporting and loading plates are shown in Fig. 3. Due to the symmetry of the tested beam, it is applicable to model only one quarter of the beam and choosing the appropriate boundary conditions to exactly simulate the whole beam. One quarter of the beam is modeled with dimensions: 50 mm width, 250 mm height, and a total length of 1500 mm. Also, one quarter of the reinforcement will be used to reinforce the modeled beam. Table shows the average material properties for the concrete used in the experimental work and are used as an input for the finite element model.

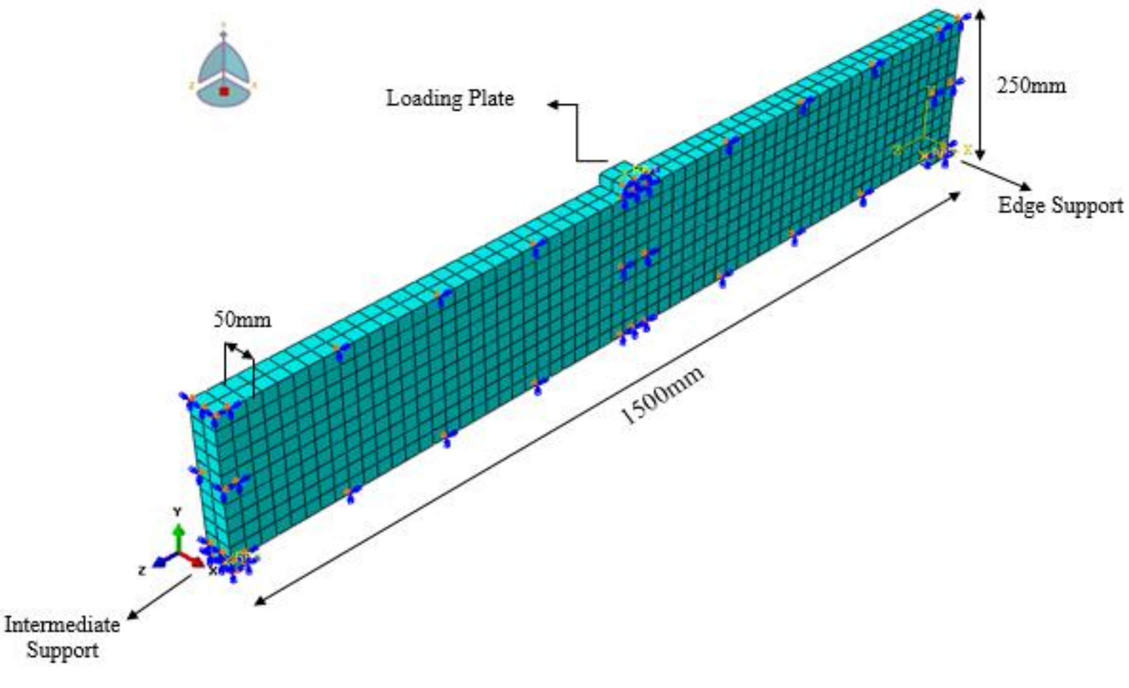

Figure 3: Geometry, mesh, and boundary conditions of the finite element beam

Table 1. Average Material Properties for Concrete

Density, kg/m^3	**2300**
Poisson's ratio	0.2
Young's modulus, GPa	27.8
Cubic compressive strength, MPa	40
Tensile strength, MPa	3.8

3.1 Elements for the beam and its reinforcement

The 3D 8-noded hexahedral (brick) elements with reduced integration (C3D8R) available in the ABAQUS library are used to model the concrete beam. 3D linear truss (T3D2) elements are used to model the steel bars. Stirrups are modeled using the same elements of the longitudinal bars (T3D2), and both types of reinforcement are embedded in the concrete geometry, so their translation degrees of freedom are constrained to the corresponding concrete degrees of freedom [8].

The beam end and middle supports, which are provided in the experimental test by the frame supporting beams, are modeled in a simplified way to avoid stress concentrations if displacement boundary conditions are applied on a line. The end and middle supports are modeled using a 50 mm wide steel plates constrained to a reference point lying on the lower surface of the plates with a rigid body motion. The vertical displacement (U2) of the reference point is blocked and all other rotational degrees of freedom are allowed. Also, two steel loading plates are used for loading the beam and are constrained to a reference point on their upper surface where the concentrated load is applied.

3.2 Tension stiffening relationship

The tensile behavior of concrete in a damaged plasticity model in ABAQUS requires a post-failure stress -strain relationship for concrete in tension and the damage parameter values. Nayal and Rasheed [9] developed a model for tension stiffening of concrete, and this model was modified for ABAQUS and used by Wahalathantri et al. [10]. Figures 4 shows the tension stiffening model proposed by [10].

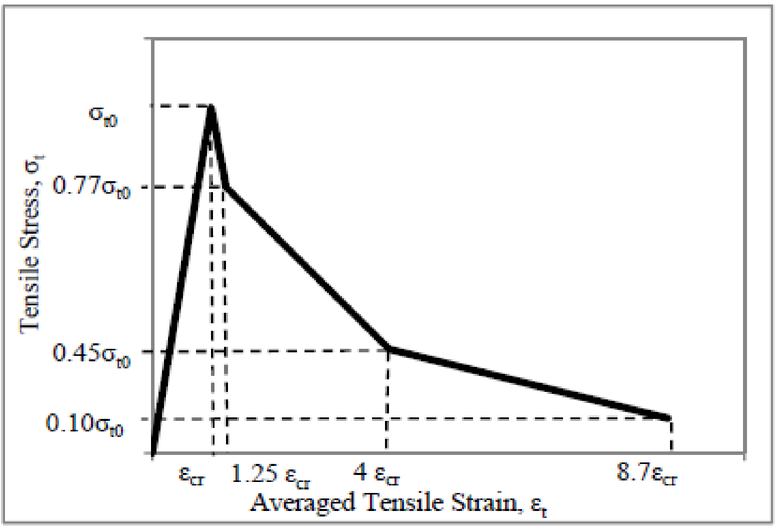

Figure 4: Wahalathantri et al. [10] tension stiffening model

The tensile behavior of concrete is characterized by a linear stress-strain relationship till reaching the value of cracking stress, where micro-cracks start to initiate. The transfer of these micro-cracks to macro-cracks are then captured using a softening stress-strain relationship which can be defined in ABAQUS using the 'tension stiffening' option. In this study, the modified tension stiffening model by [10] is used but with one modification. The model proposed by [10] was a modification to the Nayal & Rasheed's model, and one of its modifications was: the sudden drop at critical tensile strain was slanted from (ε_{cr}, σ_{t0}) to (1.25 ε_{cr}, 0.77 σ_{t0}) to avoid run time errors in ABAQUS material model. However, even after this modification, run time errors occur at high loading values. Therefore, in this study the sudden drop at critical tensile strain is again shifted from (1.25 ε_{cr}, 0.77 σ_{t0}) to (1.4 ε_{cr}, 0.77 σ_{t0}) to avoid run time errors at large load values. Figure 5 shows the tensile stress-strain relationship for the modeled concrete using both Wahalathantri et al. [10] model and the modified model for this study.

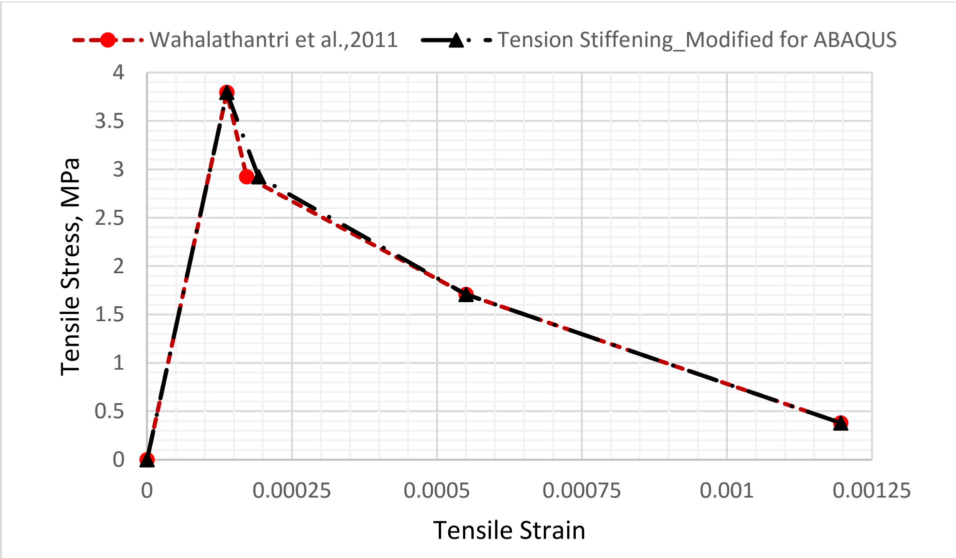

Figure 5: Modified tension stiffening model

3.3 Compressive behavior

The compressive behavior of concrete is characterized by a linear stress-strain relationship till reaching the value of yield stress, then obeys a non-linear relationship till reaching the maximum compressive strength and then a strain softening in the descending portion till failure. This can be modeled using the 'concrete compression damage' in ABAQUS. In this study the complete stress-strain relationship for concrete in compression is determined in accordance with the Hsu & Hsu model [11]. However, a modification is done at the maximum strain to escape from negative or decreasing values for plastic strain. Figure 6 shows the compressive stress-strain relationship according to Hsu & Hsu model versus the modified model used in the presented study.

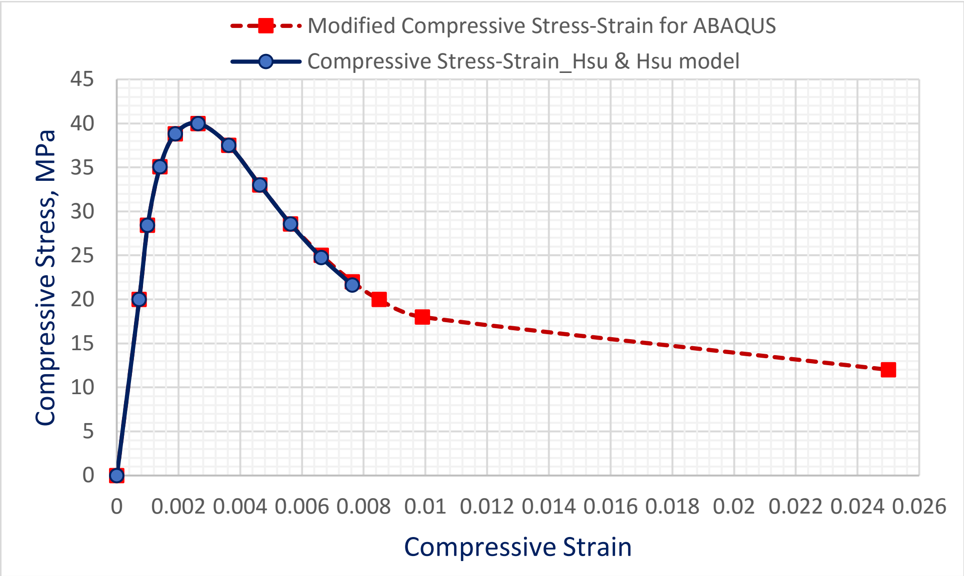

Figure 6: Compression stress-strain relationship for concrete used for ABAQUS

3.4 Damage parameters

Two damage parameters are required in a concrete damage plasticity model: one for tension, d_t, and another for compression, d_c. The tensile damage parameter, d_t, is defined as the ration of the cracking strain to the total strain. Similarly, the compressive damage parameter, d_c, is the ratio of the inelastic strain to the total strain. Table 2 shows the values of cracking and inelastic strains versus the damage parameter values.

Table 2. Damage parameters values in compression, d_c, and tension, d_t

Compression damage		**Tensile damage**	
Inelastic strain	Damage, d_c	Cracking strain	Damage, d_t
0	0	0	0
0.000275591	0.011023642	0.000054978	0.045977011
0.000675591	0.027023642	0.000412341	0.344827586
0.001175591	0.047023642	0.001058341	0.885057471
0.001905925	0.076237017		
0.002905925	0.116237017		
0.003905925	0.156237017		
0.004905925	0.196237017		
0.005905925	0.236237017		
0.006905925	0.276237017		
0.007775591	0.311023642		
0.009175591	0.367023642		
0.024275591	0.971023642		

4. Results and Discussion

Results are presented in terms of load-deflection relationship and cracking patterns for both experimental and finite element beam. Also, the stress versus strain curves for elements located in the compression and tensile zones are presented for the ABAQUS beam. The numerical beam is loaded by a concentrated load acting at the reference point coupled to the loading plate, the same as the experimental beam.

4.1 Load-deflection relationship (experimental Vs. Finite)

Figure 7 shows the load-deflection curves for the experimentally tested beam versus the ABAQUS one. The curves show a very good agreement between the experimental and finite results. Initially the two beams show the same linear behavior till reaching a load of 20 kN. After that the two beams show approximately a same non-linear relationship till reaching the failure load at 91 kN for the experimental beam and 88 kN for the ABAQUS beam.

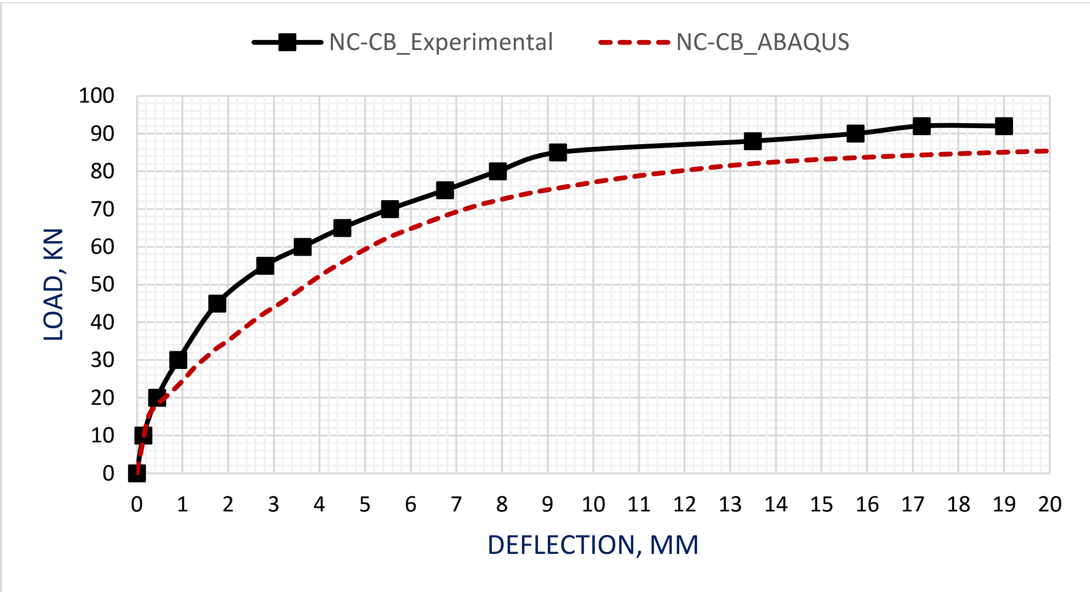

Figure 7: Load-deflection curves for the experimental Vs. the ABAQUS beam

4.2 Cracking patterns

In a concrete damage plasticity model, cracks can be graphically represented by introducing an effective crack direction, and the direction of the maximum principal plastic strain, as it is normal to the crack plane, determines the crack direction [7]. Lubliner et al. [12] assumes that cracks are initiated at points where the tensile equivalent plastic strain, and the maximum principal plastic strain are both positive. Therefore, the contour plots of the maximum principal plastic strain (PE, Max, Principle) are shown in Fig. 8 which simulate the crack initiation and propagation at critical sections during loading.

As shown in Figs. 8 and 9, the cracking patterns for the finite model and the experimental beam are nearly the same. Cracks initiated at the section of negative bending moment, at the middle support, followed by cracks propagating at sections of positive bending moments at the middle of each span at the lower surface as the load increases.

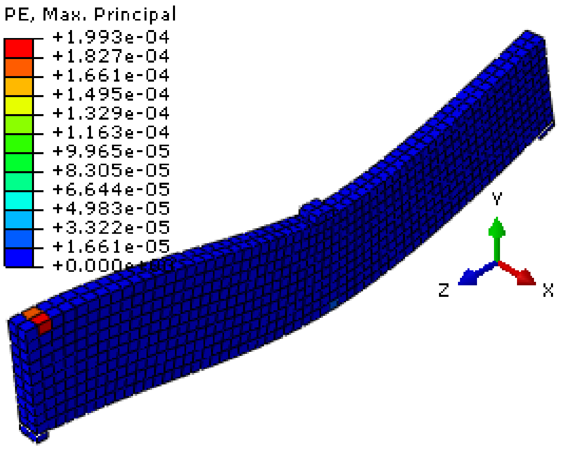

Figure 8a: Initiation of cracks at the upper face of the beam at the beginning of loading

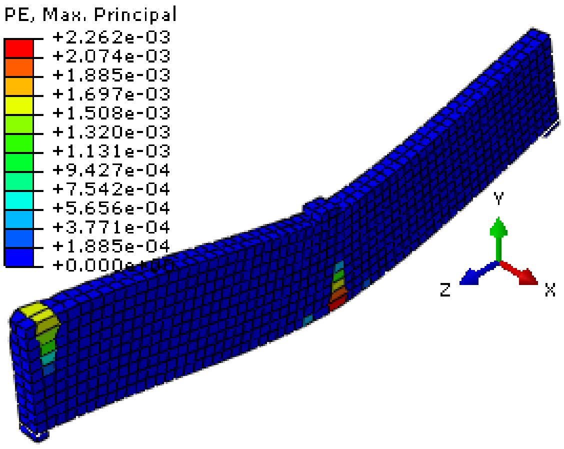

Figure 8b: Propagation of cracks at critical sections (20% P_u)

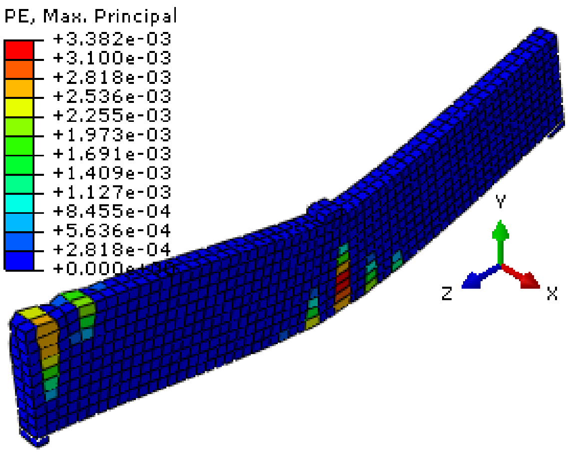

Figure 8c: Propagation of cracks at critical sections (50% P_u)

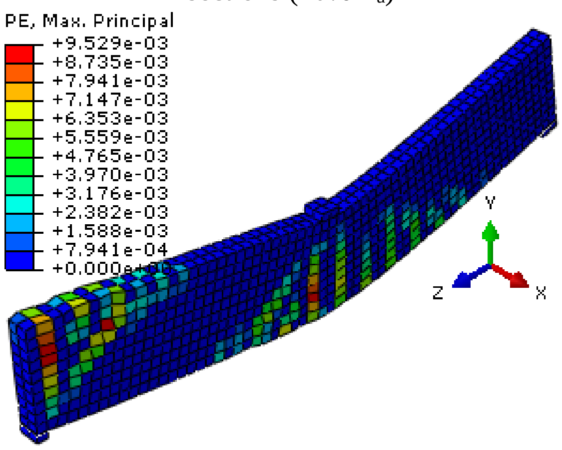

Figure 8d: Propagation of cracks at critical sections (80% P_u)

Figure 8: Predicted crack patterns for the ABAQUS beam at various levels of loading

Figure 9a: Crack patterns at the beam upper surface of the beam (-ve moment)

Figure 9b: Crack patterns at the lower surface at the middle of the right span (+ve moment)

Figure 9: Crack patterns for the experimental beam at failure load

4.3 Stresses within the model elements

Tensile and compressive stresses along the ABAQUS beam are shown in Fig. 10 as stresses contours. The contours show tensile stresses at the section of positive and negative moments. As load initiated, elements at critical sections started to sustain load till the stresses reached the maximum specified strength in the tension stiffening model. As shown in Fig. 10a, the stresses at the red colored elements are about 3.7 MPa in tension, and this is nearly the maximum strength specified. As load increased, these elements started to crack, and can't sustain tensile stresses anymore. Therefore, stresses are transferred to the adjacent elements as shown in Fig. 10b. At failure load, nearly all elements in the tensile zone of the beam were cracked, and high compressive stresses were shown directly under the point of load application. As these stresses exceeded the concrete specified compressive strength, crushing of concrete happened.

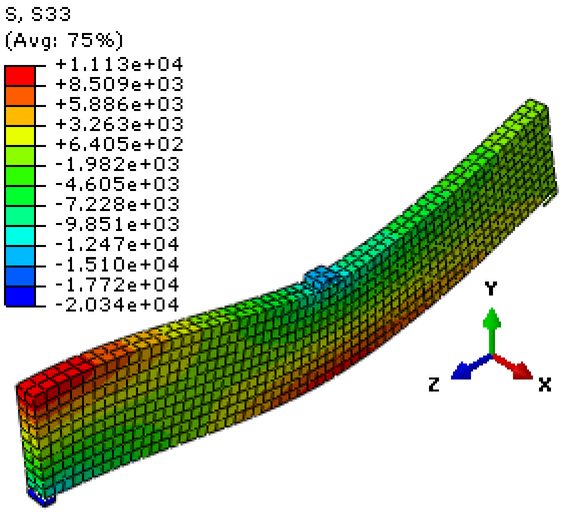

Figure 10a: S33 contours at load initiation

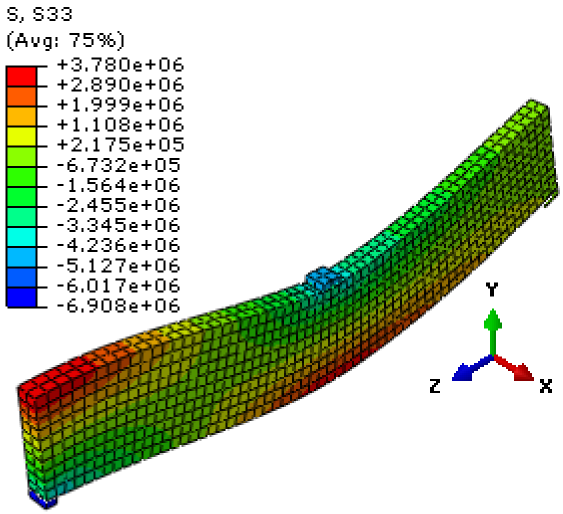

Figure 10b: S33 contours at 40% P_u

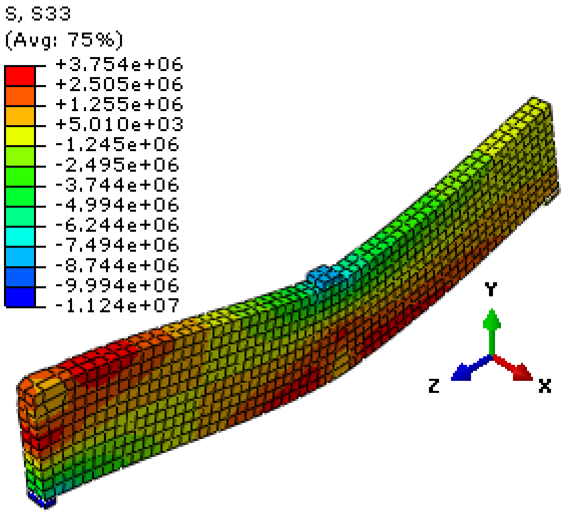

Figure 10c: S33 contours at 60% P_u

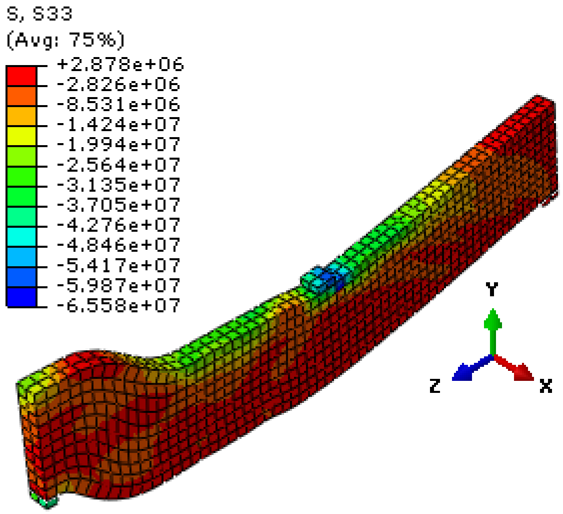

Figure 10d: S33 contours at 80% P_u

Figure 10: Stresses, *S33*, contours diagram

5. Conclusions

A finite element model for a two-span reinforced concrete beam Using the ABAQUS software package was presented. The concrete damage plasticity and the tension stiffening models together with the elastic stiffness damage parameters were introduced to the model. The model was validated using results of an experimentally tested beam and the following points were concluded from this study:

- i The concrete damage plasticity model in ABAQUS can catch the non-linear stress-strain behavior of plain concrete.
- ii The modified tension stiffening model presented in this study can be used for simulating the tensile behavior of concrete at high loading levels avoiding ABAQUS run time errors.
- iii The proposed finite element model can be used to track the cracks initiation and propagation in RC beams.

6. References

[1] NazemM, Nz.M.N, Rahimani, I., and rezaee-Pajand, M., (2009)., “Nonlinear FE Analysis of Reinforced Concrete Structures Using a Tresca-Type yield Surface”, Scientica Iranica No. 6, Vol. 16, pp. 512-519.

[2] Chaudhari, S.V., and Chakrabarti, M.A., (2012). “Modeling of Concrete for Nonlinear Analysis Using Finite Element Code ABAQUS”, International Journal of Computer Applications (0975-8887), Volume 44-No. 7.

[3] Hognestad, E., (1951). “A Study of Combined Bending and Axial Load In Reinforced Concrete Members”, ENGINEERING EXPERIMENT STATION, University of Illinois, Urbana. Bulletin Series No. 399, ENGINEERING EXPERIMENT STATION, University of Illinois, Urbana, Vol. 49, No. 22. Bulletin Series No. 399A, Vol. 49, No. 22.

[4] Bangash, M.Y.H., (1989). Concrete and Concrete structures: Numerical Modelling and Applications. London: Elsevier Science Publishers Ltd.

[5] Kmiecik, P., and Kaminski, M., (2011). “Modelling of Reinforced Concrete Structures and Composite Structures with Concrete Strength Degradation Taken into Consideration”, Archives of Civil and Mechanical Engineering, 11(3), pp.623-636.

[6] George, J., Rama JS.K., Kumar M.V.N.S., and Vasan, A., (2017). “Behavior of Plain Concrete Beam subjected to Three Point Bending using Concrete Damaged Plasticity (CDP) Model”, Materials Today: Proceedings 4, pp.9742-9746.

[7] Alrazi, E., Giulio, A., Katherine, C., and Xiangming, Z., (2017). “Nonlinear Three-Dimensional Finite-Element Modelling of Reinforced-Concrete Beams: Computational Challenges and Experimental Validation”, Engineering Failure Analysis 82, pp.92-115.

[8] Dassault Systèmes, ABAQUS 6.14 analysis user's guide, Volume V: Prescribed Conditions, Constraints and Interactions, Dassault Systèmes, USA, 2014.

[9] Nayal, R., & Rasheed, H.A. (2006). “Tension Stiffening Model for Concrete Beams Reinforced with Steel and FRP Bars”, Journal of Materials in Civil Engineering, 18(6), 831-841.

[10] Wahalathantri, Lankananda, B., Thambiratnam, D.P., Chan, T.H.T., and Fawzia, S, (2011). "A Material Model for Flexural Crack Simulation in Reinforced Concrete Elements Using ABAQUS", Proceedings of the First International Conference on Engineering, Designing and Developing the Built Environment for Sustainable Wellbeing, Queensland University of Technology, Brisbane, Qld, pp. 260-264.

[11] Hsu, L.S., & Hsu, C.-T.T. (1994). "Complete Stress-strain Behavior of High-strength Concrete Under Compression", Magazine of Concrete Research, 46(169), 301-312.

[12] Lubliner, J., Oliver, J., Oller, S., Onate, E., (1989). "A Plastic–damage Model for Concrete", Int. J. Solids Struct. 25 (3) 299–326.