EFFECT OF VISCOSITY PARAMETER ON THE NUMERICAL SIMULATION OF REINFORCED CONCRETE DEEP BEAM BEHAVIOR

*[1]Aydin Demir, [1]Hakan Ozturk, [2]Kemal Edip, [2]Marta Stojmanovska, [2]Aleksandra Bogdanovic

[1]Faculty of Engineering, Department of Civil Engineering, Sakarya University, Sakarya, Turkey
[2]Institute of Earthquake Engineering and Engineering Seismology, Ss. Cyril and Methodius University, Skopje, Macedonia

aydindemir@sakarya.edu.tr

Abstract:In the study, a parametric nonlinear finite element (FE) study is performed in order to investigate sensitivity of viscosity parameter on numerical simulation of RC deep beam behavior. For this objective; a numerical verification of an existing experimental study of a deep beam is conducted by using ABAQUS software with varied values of viscosity parameter. Results of the experimental and numerical studies are compared in terms of load-displacement behaviors, and rate of convergence and calculation time of the numerical models. Selection of an optimum viscosity parameter and its definition to FE model improves significantly performance of convergence and reduces analysis time of numerical simulations of RC deep beams.

Key words: Viscosity parameter, concrete damage plasticity, finite element analysis, ABAQUS, reinforced concrete deep beam

Introduction

It is known that behavior of a structure relies on behavior of its members such as columns, beams etc. Therefore investigation of nonlinear behavior of reinforced concrete (RC) members under static and dynamic loads is very crucial to design safe structures (Caglar et al., 2015a, 2015b)

Deep beams are defined in ACI 318-14 (2014) code as RC members that are loaded on one face and supported on the opposite face. Moreover they have a clear span not exceeding four times the overall member depth.

There are several studies demonstrating that finite element (FE) analysis including an accurate numerical modeling technique and appropriate constitutive material models is also a quite reliable and robust tool to simulate and investigate nonlinear behavior of RC members (Demir et al., 2016a). Because of this reason, FE analysis is widely preferred tool by researchers for their scientific researches (Edip et al., 2013).

Two different constitutive material models are offered in ABAQUS/Standard, which is an implicit analysis program, for the analysis of concrete at low confining pressures: the smeared crack concrete model and the concrete damaged plasticity (CDP) model. Moreover, the CDP model is based on the assumption of isotropic damage and considers the degradation of the elastic stiffness induced by plastic straining both in compression and tension. For the CDP material model it can be defined flow potential, yield surface, and viscosity parameters ("ABAQUS online documentation server").

In implicit analysis programs, constitutive models lead to severe convergence difficulties due to softening behavior and stiffness degradation of materials. The use of a viscoplastic regularization of the constitutive equations is one of common technique to overcome some of these convergence difficulties ("ABAQUS online documentation server"). For the CDP material model in ABAQUS software, viscoplastic regularization can be taken into account by defining a viscosity parameter in material definitions.

Ren et al. (2015) conducted a finite element study of PPC bridge deck panels based on validation of an experimental test using concrete damage plasticity model. In a part of the numerical study they made a parametric analysis with varied values of viscosity parameter. Numerical results demonstrated that lower values of viscosity parameter increased calculation accuracy and increased the calculation time.

Moreover, Ma et al. (2012) studied the numerical simulation of an existing experimental study on the energy absorption columns. ABAQUS software was used to perform numerical research. The concrete damage plasticity model was adopted for the calculation of constitutive concrete material used in the columns and the viscosity coefficient was discussed. It was specified that until finding a reasonable value for viscosity parameter, a parametric study should be conducted in order to improve convergence of numerical simulation.

No default value for viscosity parameter is proposed in ABAQUS manuals and literature in which it is however highly recommended to be defined to FE models to overcome convergence difficulties in numerical simulations. Because of that reasons, the motivation of the study is to investigate sensitivity of viscosity parameter on nonlinear numerical simulation of RC deep beams. For this purpose numerical verification of an existing experimental study of a deep beam is performed by employing a commercial finite element software ABAQUS. Results of the experimental and numerical studies are compared in terms of load-displacement behaviors, and rate of convergence and time of numerical models.

Viscosity parameter (μ)

The CDP model can be regularized in ABAQUS/Standard using viscoplasticity by permitting stresses to be outside of the yield surface. Viscoplastic regularization of the constitutive equations causes the consistent tangent stiffness of the softening material to become positive for sufficiently small time increments. Generalization of the Duvaut-Lions regularization is used according to which the viscoplastic strain rate tensor, $\dot{\varepsilon}_v^{pl}$, is defined as (Eq. 1);

$$\dot{\varepsilon}_v^{pl} = \frac{1}{\mu}\left(\varepsilon^{pl} - \varepsilon_v^{pl}\right) \tag{1}$$

Where μ the viscosity parameter representing the relaxation time of the viscoplastic system, and ε^{pl} is the plastic strain evaluated in the inviscid backbone model.

Additionally, for the viscoplastic system, a viscous stiffness degradation variable, d_v, is defined as

$$\dot{d}_v = \frac{1}{\mu}(d - d_v) \tag{2}$$

Here d is the degradation variable evaluated in the inviscid backbone model. The stress-strain relation of the viscoplastic model is defined as follows;

$$\sigma = \frac{(1-d_v)D_0^{el}}{\varepsilon-\varepsilon_v^{pl}} \tag{3}$$

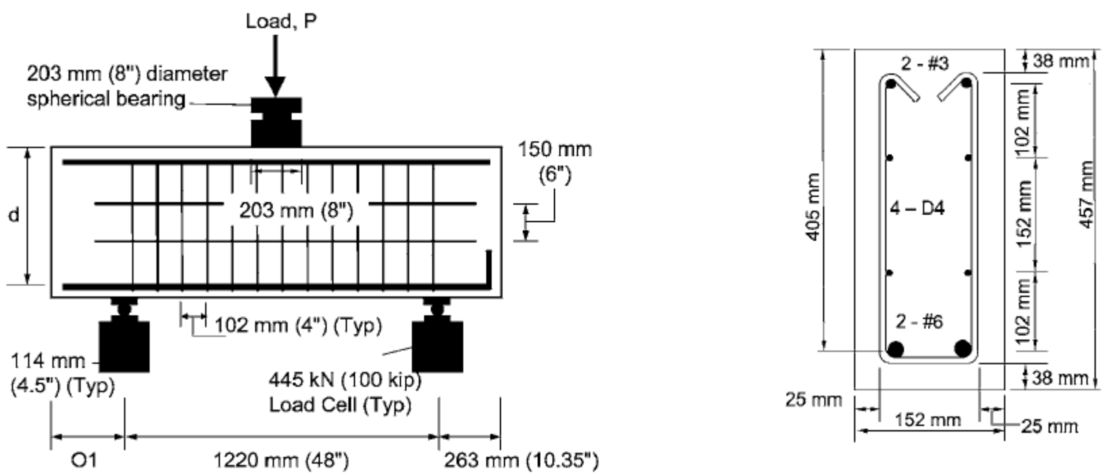
Taking a small value for the viscosity parameter, small adequate compared to the characteristic time increment, usually contributes to improve the rate of convergence of the model in the softening regime, without compromising numerical results. While $t/\mu \to \infty$, the solution of the viscoplastic system relaxes to that of the inviscid case, where t is time.

Value of the viscosity parameter can be defined as part of the concrete damaged plasticity material behavior definition. The default value of μ is zero in ABAQUS/Standard that corresponds to omission of viscoplastic regularization. If μ differs from zero, output results of the stiffness degradation and plastic strain refer to the viscoplastic values, d_v and ε_v^{pl} respectively ("ABAQUS online documentation server").

Experimental study

An experimental study conducted by Roy and Brena (2008) are selected as a reference study in order to create a numerical model of an RC deep beam. One of the specimens named as "DB1.0-0.75L" in the reference study is chosen as a reference verification specimen. The ratio of shear zone to effective depth of the section (a/d) is given as 1.0.

The details of specimen geometry, reinforcement, and experimental test setup are displayed in Fig. 1. Additional details about test setup, loading procedure and material properties can be found in the relevant study.



a) Side view b) Cross Section

Figure 1. Specimen geometry, reinforcement, and test setup (Roy & Breña, 2008)

Load-displacement result of the test is presented in Fig. 2. Since no tabular data for load-deflection graph were given in the reference study, loads and corresponding displacement values are determined manually (Demir et al., 2016b).

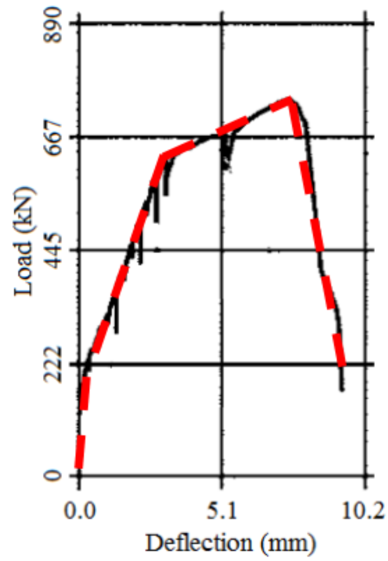


Figure 2. Load-deflection result of the test (Demir et al., 2016b)

Numerical modeling

In the study, the FE-code ABAQUS/Standard (ABAQUS/CAE 6.13-2 SE) is used to create numerical simulation. Inelastic behavior of concrete is defined to FE model by using Concrete Damaged Plasticity (CDP) model. Similar numerical FE model of a deep beam proposed in the study of Demir et al. (2016b) is used to conduct the present numerical parametric study on the viscosity parameter. Because of that reason, FE modeling procures to create current numerical simulation are not explained here and whole details can be found in the related study. Moreover FE model and verification result with test result are demonstrated in Fig. 3.

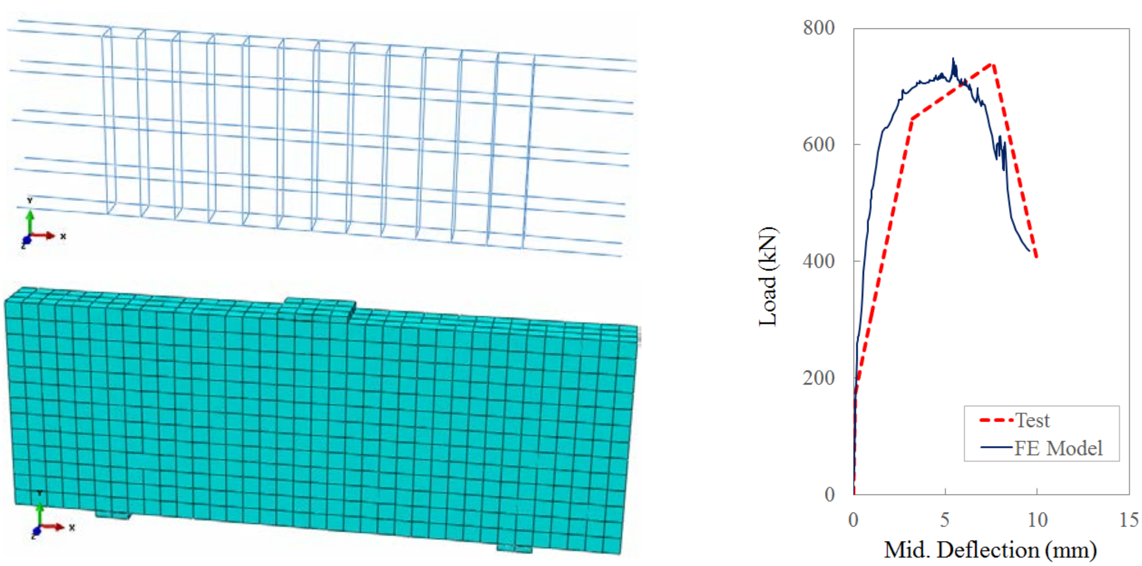


Figure 3. Meshed FE model and and verification result (Demir et al., 2016b)

Parametric study

In order to investigate effect of viscosity parameter on numerical behavior of RC deep beam, a parametric study is performed. For that objective 8 different numerical models are created with different viscosity parameter, tabulated in Table 1. Results of the experimental and numerical studies are compared in terms of load-displacement behaviors and ultimate load levels. Additionally, total number of iterations to finish the numerical analysis, percentage of convergence according to step time (taken as 1 second in the FE model), and comparison of error of numerical results with the test result in terms of ultimate load levels are tabulated.

Results and discussion

The test and numerical study results are tabulated in Table 1 and plotted in Fig. 4. Results of the total number of iterations to finish the FE analysis and percentage of convergence according to step time are specified. Moreover, ultimate load (f_u) values of the test and numerical models are given, and error of numerical results in f_u are compared with the test result in the table.

It can be clearly seen from Fig. 4 that viscosity parameter plays very important role on numerical results in a way that it changes significantly the numerical load-displacement behavior of deep beams. However load-displacement graphs could not be obtained for models DB-1 and DB-2 because the FE models did not converged. The FE model (DB-1) aborted with very small percentage of convergence (6 %) under value of viscosity parameter, zero which is a default value of ABAQUS software. Moreover with the definition of a very small viscosity parameter to the FE model, DB-2, the simulation similarly did not converged but the percentage of convergence has slightly increased (14 %). Due to nonconvergent results, duration of analysis (total number of iteration) could not be measured for that FE models.

Table 1. Results of the experimental and numerical study

#	Name of Model	μ	Total no. of iteration	Convergence %	f_u (kN)	Error % in f_u
1	DB-Test	-	-	-	740	-
2	DB-1	0	n/a	6	n/a	n/a
3	DB-2	0.00001	n/a	14	n/a	n/a
4	DB-3	0.00005	605	100	692	-6.5
5	DB-4	0.00010	508	100	697	-5.8
6	DB-5	0.00050	444	100	738	-0.3
7	DB-6	0.00100	341	100	786	6.3
8	DB-7	0.00500	305	100	929	25.5
9	DB-8	0.01000	278	100	1109	49.9

Along with the increase in value of μ, numerical models have started to converge. For the models, DB-3 and DB-4, the numerical results are very similar to the test results in terms of load-displacement behavior of the tested deep beam. Percentage of error in ultimate load level stayed under 10 % as well. Total number of iterations for DB-3 and DB-4 are 605 and 508 respectively.

With increase in value of viscosity parameter (above 0.0005), numerical load-displacement behaviors of the models of DB-5 through DB-8 have started to lose their fitness in terms of ultimate load levels. Especially when the value of μ is above 0.005, the models of DB-7 and DB-8 performed very poor behavior and the results substantially deviated from test results. However total number of iterations decreased significantly.

Conclusions

In the present study, a parametric FE study was performed in order to investigate sensitivity of viscosity parameter on numerical simulation of RC deep beam behavior. For this objective; a numerical verification of an existing experimental study of a deep beam is performed by using ABAQUS software with varied value of viscosity parameter assigned to the FE models. Results of the experimental and numerical studies are compared in terms of load-displacement behaviors, and rate of convergence and time of numerical models.

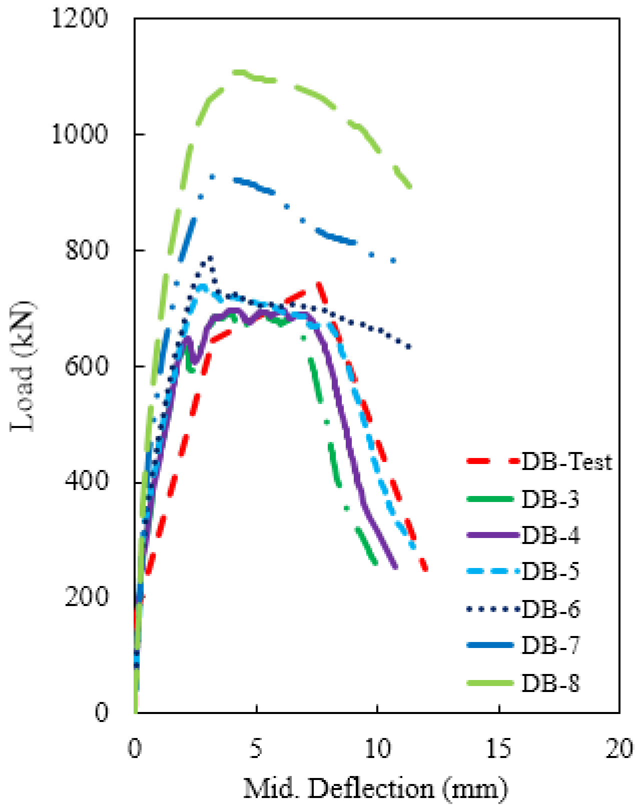


Figure 4. Load-displacement results

The numerical results are deduced that omission of viscoplastic regularization by taking the default value of μ as zero in ABAQUS generally causes nonconvergent results in FE simulations. Viscosity parameter however plays very important role on numerical simulation of RC deep beams. Along with definition of a viscosity parameter to the simulation, not only increase in convergence performance of FE model but also a significant reduce in duration of an analysis can be obtained.

For numerical simulation of RC deep beams, an optimum value for viscosity parameter in terms of numerical convergence and analysis time should be selected and assigned to the FE model. The optimum value of the parameter can be taken as 0.0005 giving very accurate numerical result in terms of load-displacement behavior. Above that value, numerical results become distant from the test results. Nevertheless, it should be noted that until finding a reasonable viscosity parameter, a parametric numerical sensitivity study should be conducted with varied values of the parameter in order to improve calculation accuracy of numerical simulation of RC deep beams.

References

ABAQUS/CAE 6.13-2 SE. Providence, RI: Dassault Systèmes Simulia Corp.

ABAQUS online documentation server. Retrieved from http://50.16.225.63

ACI 318-14. (2014). *Building Code Requirements for Structural Concrete*. American Concrete Institute.

Caglar, N., Demir, A., Ozturk, H., & Akkaya, A. (2015a). A new approach to determine the moment-curvature relationship of circular reinforced concrete columns. *Computers and Concrete*, *3*, 321–335. http://doi.org/10.12989/cac.2015.15.3.321

Caglar, N., Demir, A., Ozturk, H., & Akkaya, A. (2015b). A simple formulation for effective flexural stiffness of circular reinforced concrete columns. *Engineering Applications of Artificial Intelligence*, *38*, 79–87. http://doi.org/10.1016/j.engappai.2014.10.011

Demir, A., Caglar, N., Ozturk, H., & Sumer, Y. (2016a). Nonlinear finite element study on the improvement of shear capacity in reinforced concrete T-Section beams by an alternative diagonal shear reinforcement. *Engineering Structures*, *120*, 158–165. http://doi.org/10.1016/j.engstruct.2016.04.029

Demir, A., Ozturk, H., & Dok, G. (2016b). 3D Numerical Modeling of RC Deep Beam Behavior by Nonlinear Finite Element Analysis. *Disaster Science and Engineering*, *2*(1), 13–18. Retrieved from http://www.disasterengineering.com/article/view/5000183438

Edip, K., Garevski, M., Butenweg, C., Sesov, V., Bojadjieva, J., & Gjorgjiev, I. (2013). Boundary effects on seismic analysis of multi-storey frames considering soil structure interaction phenomenon. In *International Conference on Seismic Design of Industrial Facilities*, *RWTH Aachen University*. RWTH Aachen University.

Ma, H., Wang, H. H., Li, Z. B., Sun, X. Y., & Zhang, X. W. (2012). The Finite Element Analysis of RC Energy Absorption Columns Based on ABAQUS Software. *Applied Mechanics and Materials*, *174–177*, 2122–2125. http://doi.org/10.4028/www.scientific.net/AMM.174-177.2122

Ren, W., Sneed, L. H., Yang, Y., & He, R. (2015). Numerical Simulation of Prestressed Precast Concrete Bridge Deck Panels Using Damage Plasticity Model. *International Journal of Concrete Structures and Materials*, *9*(1), 45–54. http://doi.org/10.1007/s40069-014-0091-2

Roy, N. C., & Breña, S. F. (2008). Behavior of deep beams with short longitudinal bar anchorages. *ACI Structural Journal*, *105*(4), 460–470.

