Comparative Analysis of Constitutive Models for Concrete Using the Finite Element Method: Microplane Model and Concrete Damaged Plasticity

Vinicius Vaz[1], Luan Reginato[1], Adriana Elisabete Limberger[1], Luiz Fernando Zubeldia Dal Posso[1]

[1]Itaipu Parquetec, Centro Avançado em Soluções para Barragens (CEASB)
Avenida Presidente Tancredo Neves, 6731, CEP 85867-900, Foz do Iguaçu, Paraná/Brasil.
viniciusvazpti@gmail.com, luan.reginato@pti.org.br, adriana.limberger@pti.org.br, luiz.zubeldia@pti.org.br

Abstract. The modeling of reinforced concrete structures has significantly advanced in Computational Mechanics, allowing for the simulation of complex phenomena such as cracking and crushing in concrete structures. With the Finite Element Method (FEM), it has become possible to develop more sophisticated models that describe the behavior of concrete more realistically through plastic constitutive models with associated damage. This paper aims to compare the use of two constitutive models available in the Ansys and Abaqus software: the Microplane Model and Concrete Damaged Plasticity, respectively. A reinforced concrete beam was simulated, and two key parameters from each model were varied to investigate changes in the element's failure load. The damage distribution in the tensioned regions was also analyzed and compared with the observed cracking pattern of the concrete beam. The results indicate that Abaqus is better suited for simulating reinforced concrete structures, providing more accurate predictions of damage and failure patterns.

Keywords: Concrete, Microplane, Concrete Damaged Plasticity, Ansys, Abaqus.

1 Introduction

Reinforced concrete is crucial in modern infrastructure construction, combining the compressive strength of concrete with the tensile strength of steel. According to Mehta and Monteiro [1], concrete is highly valued for its robustness, making it a popular material choice among structural engineers. However, despite the numerous engineering practices employed to ensure a high-quality product, it is essential to consider the long-term durability of concrete.

Design criteria for reinforced concrete structures are established by technical standards, such as the Brazilian standard ABNT NBR 6118 and the American ACI 318, which consider analytical methods and basic assumptions about the behavior of materials and structural elements. With the rapid advancement of technology and materials, concrete structures with different purposes are being created, requiring more refined methods for analyzing their structural behaviors [2]. Consequently, the application of numerical simulation based on nonlinear analysis and the finite element method (FEM) has become an important tool for designing concrete structures, considering both the physical nonlinearity of materials and the geometric nonlinearity of elements.

Kaklauskas et al. [3] highlight that numerical simulation is an effective solution for dealing with geometric limitations and the complexity of boundary conditions. As pointed out by Gribniak et al. [4], standard methods ensure the safety of designs but do not accurately reflect the actual behavior of stresses and strains in structural elements. Thus, numerical simulation of the behavior of reinforced concrete elements, based on the Finite Element

Method (FEM), requires approximations and optimizations to achieve results that closely resemble reality [5]. The effectiveness of nonlinear finite element analyses depends on the decisions made during modeling, including the choice of constitutive models, parameters, boundary conditions, loadings, and finite element sizes.

Developing constitutive equations that incorporate all material characteristics is a complex challenge. In recent decades, numerous studies have been conducted on modeling the behavior of concrete. Most constitutive models recreate macroscopic stress-strain relationships under various loading conditions, often neglecting the microscopic mechanisms of behavior. At a macroscopic level, models can be classified based on elasticity theory, plasticity theory, and fracture and damage mechanics. The choice of the appropriate constitutive model depends on the type of problem to be analyzed. For the analysis of reinforced concrete elements, models based on elasticity or plasticity theory can provide good results due to the stress redistribution caused by the presence of reinforcement.

The main objective of this study is to perform a comparative analysis of concrete constitutive models, specifically the Microplane Model available in Ansys and Concrete Damaged Plasticity available in Abaqus. Using computational simulations based on the Finite Element Method (FEM), we aim to better understand the mechanical behavior of reinforced concrete under the same loading conditions by varying some variables within each constitutive model. Through these simulations, the behaviors of concrete were observed and analyzed using representative and comparative graphs, providing a view of the differences and similarities between the two constitutive models.

2 Constitutive Models

2.1 Microplane (Ansys)

To more precisely describe anisotropy in concrete, an approach based on microplane theory was developed, proposed by Bazant and Gambarova [6] and Bazant and Oh [7]. This approach treats each material point as a set of microplanes oriented in various directions, positioned on the surface of a unit sphere centered at the point in question. Deformations on these microplanes are calculated by applying kinematic and/or static constraints to the macroscopic strain tensor. From the stress-strain relationships specific to each microplane, the stresses on each microplane are determined. By enforcing energy equivalence conditions, the macroscopic stress state can be obtained, allowing for the evaluation of the material’s stiffness degradation.

2.2 Concrete Damage and Plasticity (Abaqus)

The Concrete Damaged Plasticity (CDP) model, initially developed by Lubliner et al. [8] and later refined by Lee and Fenves [9], is based on the concepts of plasticity and damage mechanics to calculate the material's stiffness degradation. Although it was specifically designed for concrete, SIMULIA [10] notes that the model can also be applied to other materials with quasi-brittle behavior.

In the context of structural engineering, the term "damage" refers to the reduction in stiffness caused by the development of cracks in concrete under loading. The propagation of these cracks results in a decrease in the material's stiffness and load-bearing capacity. In the Concrete Damaged Plasticity (CDP) model, damage is represented by scalar variables known as damage parameters, with *dc* used for compression and *dt* for tension.

3 Reference Experimental Test

The reference test used for the computational simulation was described in the article "Numerical and Experimental Analysis of Deflection Evolution in Reinforced Concrete Beams under Repeated Cyclic Loading" [11]. This paper aims to describe the structural behavior of reinforced concrete beams subjected to cyclic loading through numerical and experimental analyses, with the primary objective of quantifying the deflection growth of the beams under repetitive load cycles. The experiment involved reinforced concrete beams with flexural reinforcement and stirrups, with a rectangular cross-section. The beams were simply supported at the ends, and the loading was applied at the third points (Figure 1).

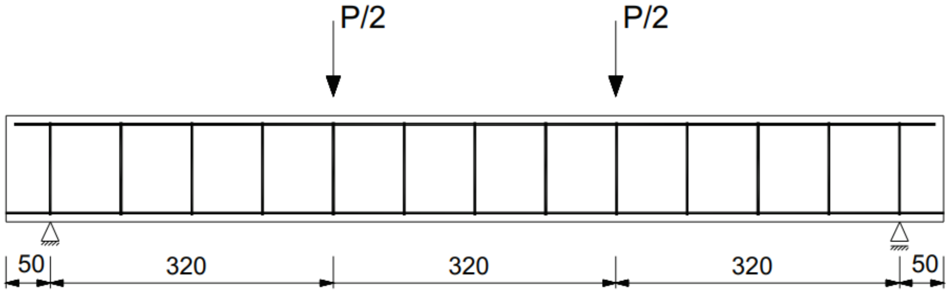


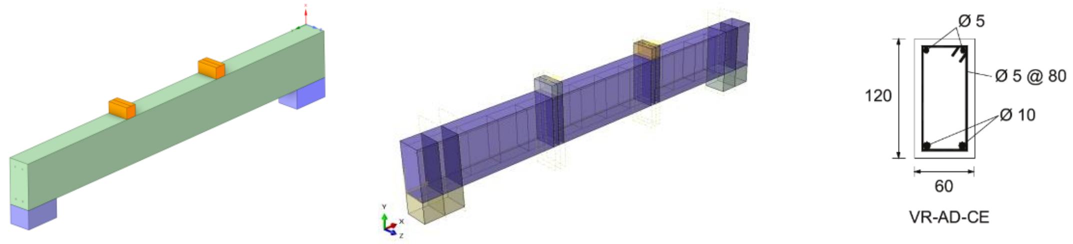
Figure 1: Dimensions and supports of the reference beam.

In the tests, the goal was to determine the loss of stiffness in the beams resulting from the application of load cycles, by measuring vertical displacements and deformations in the reinforcement and in the compressed concrete. In the concrete characterization tests, the following values were obtained: average compressive strength of 40.13 MPa and average tensile strength (obtained from the diametral compression test) of 3.20 MPa.

4 Numerical analysis

This paper aims to compare the use of concrete constitutive models present in the Ansys and Abaqus software, specifically the Microplane Model and CDP, respectively. In this study, the parameters D and BETA T of the Microplane Model and the dilatancy angle and fracture energy (Gf) of the CDP were varied.

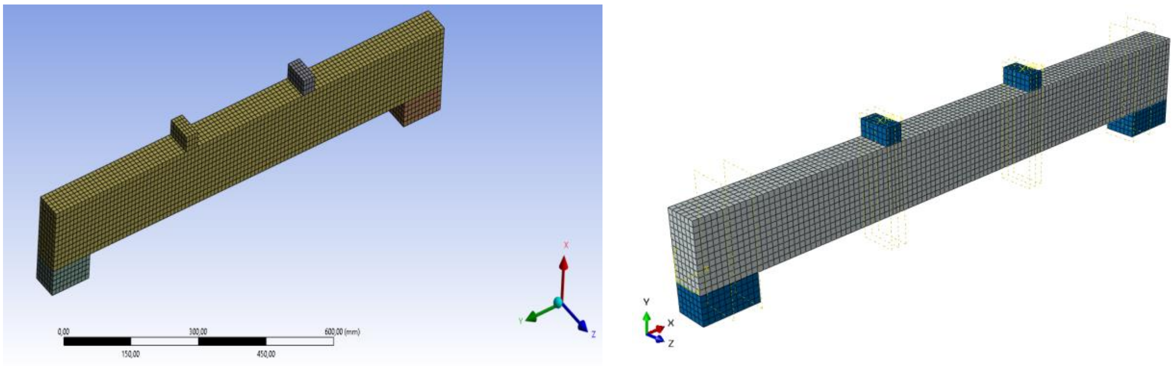
The geometry used for the numerical simulations was the same as that used in the reference article. Figure 2 shows the 3D model in Ansys (Figure 2.a) and Abaqus (Figure 2.b). Figure 2.c displays the profile of the rectangular section of the analyzed beam along with the dimensions of the reinforcement and stirrups. The orange blocks on the top of the concrete block represent the displacement application points on the structure, each with a width of 50 mm. The rectangular concrete section has dimensions of 120 mm x 60 mm with a length of 1060 mm.



a) b) c)

Figure 2: Beam modeling in a) Ansys and b) Abaqus; c) Cross-section of the reference beam VR-AD-CE.

The mesh chosen for this study was of the linear hexagonal type, with element CPT215 in Ansys and C3D8 in Abaqus, with a size of 10 mm. It had 12,573 nodes and 9,164 elements in the Ansys model and 12,008 nodes and 9,380 elements in Abaqus. Figure 3 shows the discretization of the beams in both software programs. For the supports at the bottom, one fixed support and one movable support were considered, with a width of 100 mm. Table 01 summarizes the main materials and their mechanical strength values used in the simulation.



a) b)

Figura 3: Mesh of the geometry: a) Ansys, b) Abaqus.

Tabela 1: Characteristics of the materials used in the numerical analysis.

Material	E (MPa)	Poisson	f_k (MPa)
Reinforcement	194000	0,3	515
Concrete	30421,72	0,2	40,13 (fc) 3,20 (ft)
Suport blocks	200000	0,3	250

5 Results

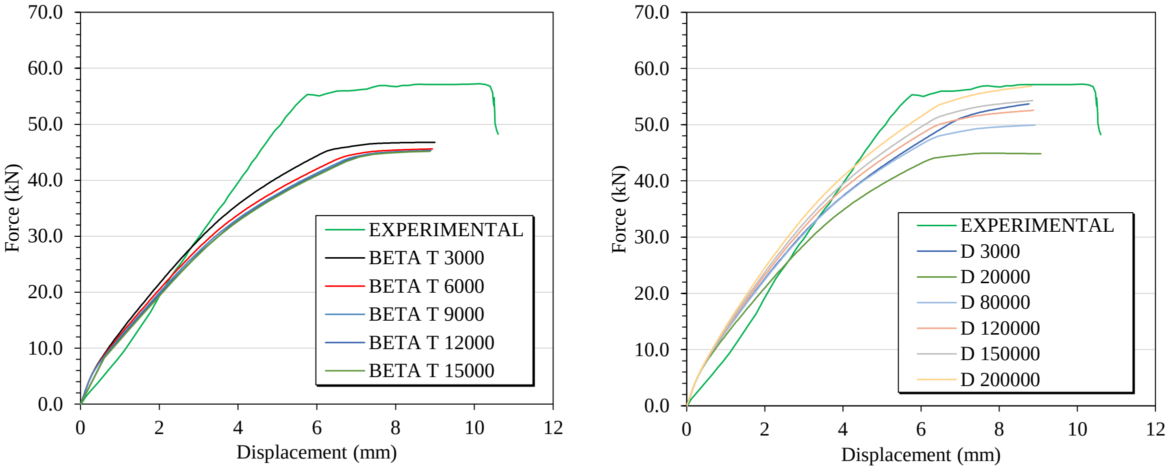
The Microplane model is implemented using APDL (ANSYS Parametric Design Language) commands, which allows for the insertion of commands through algorithms to incorporate constitutive models not available in Ansys Workbench. The APDL code for the Microplane model includes a total of 13 variables that can influence the final behavior of the concrete. Each variable can be adjusted to create numerous combinations of potential mechanical behaviors of the concrete in APDL. Thus, an initial code was developed as a starting point for modifications. Subsequently, tests were conducted by varying some variables to graphically observe the final results and make comparisons based on these outcomes. Figure 4 shows the standard APDL code used.

```
/PREP7
et,matid,CPT215 !define matid to CPT215
KEYOPT,matid,18,2
MP,EX,matid,30421.72 ! Define Elasticity Modulus
MP,NUXY,matid,0.2 ! Define Poisson's ratio
TB,MPLA,matid,,,DPC !Define Drucker-Prager
TBDATA,1,40.13,46.55,3.2,1,150000,-35 !fuc,fbc,fut,Rt,D,sigVc
TBDATA,7,2,0,2e-5,3000,1000 !R,gamt0,gamc0,betat,betac
TB,MPLA,matid,,,NLOCAL
TBDATA,1,1600,2.5 !nonlocal interaction range c, over nonlocal parameter m

! Replace SOLID185 with CPT215
allsel
esel,s,ename,,SOLID185
emodif,all,type,matid
allsel
! Print out the result:
etlist,all
/SOLU
OUTRES,ALL,ALL
```

Figure 4: APDL code for the Microplane constitutive model.

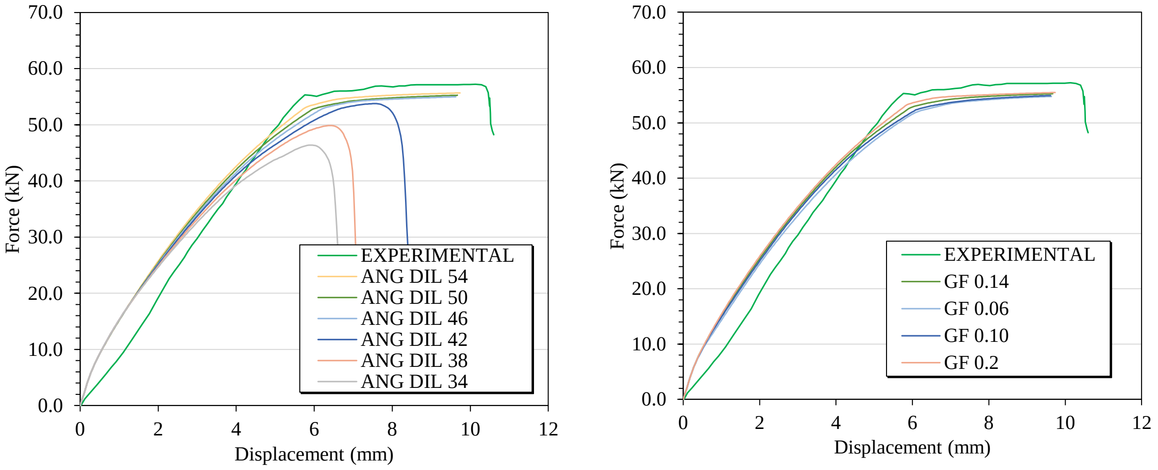
Based on the standard APDL code, variations were made to two variables in the APDL command: "Betat" and "D". These variations were made independently to represent the behavior of concrete under tension, as concrete is more susceptible to tensile failure than to compression in the studied element. In the first test, all other variables in the APDL code were kept constant, and only the "Betat" variable was changed at intervals of 3,000, 6,000, 9,000, 12,000, and 15,000. The graph of the "Betat" variation shows that increasing this variable decreases the beam's load-bearing capacity (Figure 5.a). In the second test, using the same initial APDL code, the "D" variable was changed at intervals of 3,000, 20,000, 80,000, 120,000, and 150,000. The graph of the "D" variation shows that increasing this variable increases the beam's load-bearing capacity (Figure 5.b).



a) b)

Figure 5: Force vs Displacement, varying a) BETA T and b) D of the Microplane Model

For the CDP, it was necessary to define five plastic parameters, two uniaxial stress-strain curves for compression and tension, and two inelastic damage-strain curves for compression and tension. In this paper, the same parameters and curves used in Reginato et al. [12] were employed, with variations in the dilatancy angle at values of 34º, 38º, 42º, 46º, 50º, and 54º, and fracture energy at values of 0.06, 0.10, 0.14, and 0.20 N/mm. Figure 6.a shows the results of varying the dilatancy angle, where it is observed that a lower dilatancy angle resulted in shear failure of the concrete, while a higher angle led to flexural failure. Figure 6.b presents the variation of fracture energy, with values given in N/mm, showing minimal variation in the beam's response with changes in fracture energy.

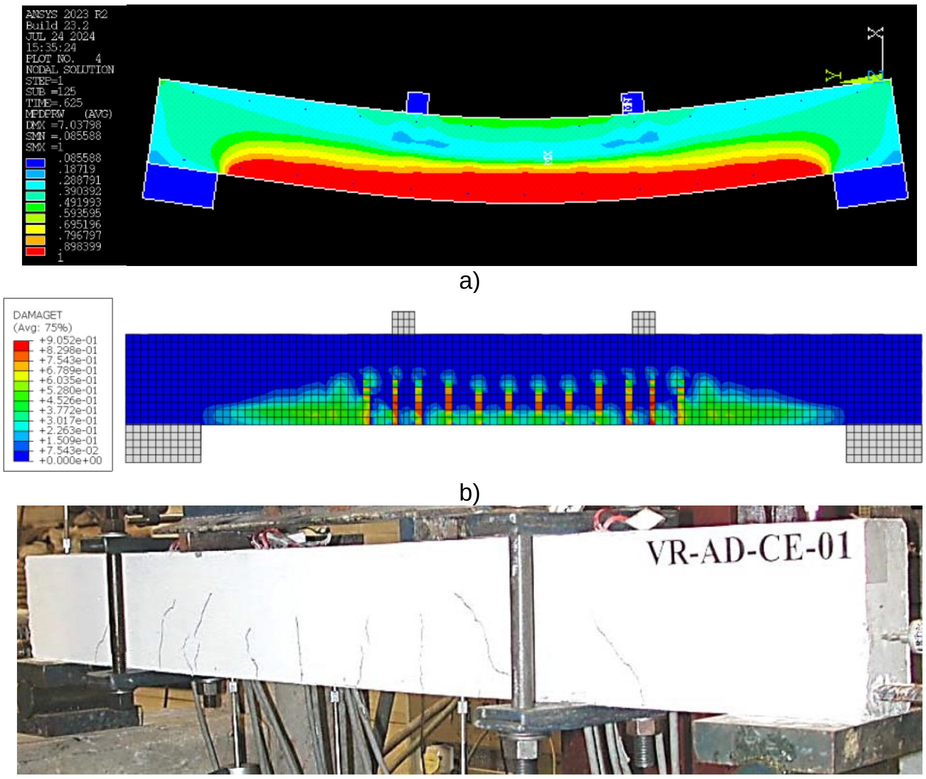


a) b)

Figure 6: Force vs Displacement, varying a) Dilatancy angle and b) Fracture energy of the CDP

Figure 7.a-b shows the damage parameter close to the failure load, with Figure 7.a in Ansys and Figure 7.b in Abaqus. It is observed that damage in the Microplane model is more dispersed compared to the damage in the CDP, which is concentrated in a few finite elements. The damage distribution in the CDP is closer to the cracks

observed in the experimental beam, as shown in Figure 7.c.



a)

b)

c)

Figure 7: Damage parameter in a) Ansys and b) Abaqus; c) Cracking in the concrete beam

6 Conclusion

This study conducted a numerical analysis of a reinforced concrete beam using two constitutive models available in Ansys and Abaqus software: the Microplane Model and Concrete Damaged Plasticity (CDP), respectively. A comparative analysis was performed by varying two parameters that define the nonlinear behavior of concrete.

In the pre-processing stage, geometry modeling in Ansys proved to be more intuitive compared to Abaqus, requiring fewer steps for geometry creation. Incorporating constitutive models presents a distinct challenge: the Microplane Model in Ansys is implemented exclusively via APDL (ANSYS Parametric Design Language), complicating its use. Conversely, the CDP in Abaqus is natively integrated into the software. Furthermore, the CDP allows the assignment of various stress-strain curves based on equations and experimental data, whereas the Microplane Model necessitates parameter calibration to fit uniaxial tests. The Microplane Model requires calibration of a greater number of parameters compared to the CDP, making the calibration process more complex. However, the Microplane Model includes a “non-local” variable that ensures mesh independence, which is absent in the CDP. On the other hand, the CDP features a viscosity parameter that facilitates processing and aids in the convergence of nonlinear problems. Additionally, the CDP supports multiple damage curves for both compression and tension, a feature not available in the Microplane Model.

In the post-processing stage, Ansys Workbench offers a more user-friendly interface. However, as the Microplane Model is implemented through APDL, result visualization from this model is constrained to APDL, complicating result analysis and extraction. Abaqus provides a more intuitive visualization of the CDP results due to its native integration.

In summary, the results and software manipulation indicate that Abaqus is better suited for the simulation of reinforced concrete structures.

5 References

[1]MEHTA, P. Kumar; MONTEIRO PAULO, J. M. Concrete, microstructure, and materials". MicGraw-hill, 2006.

[2]Stramandinoli, R. S. B.; La Rovere, H. L. (2017) Finite element model for nonlinear analysis of reinforced concrete beams and plane frames. Revista IBRACON de Estruturas e Materiais, v. 10, n. 2, p. 386-414–DOI: 10.1590/S1983 -41952017000200007.

[3]Kaklauskas, G.;Gribniak, V.;Girdzius, R. (2011) Average stress-average strain tension-stiffening relationshipsbased on provisions of design codes. Journal of Zhejiang University-SCIENCE A, v 12, p. 731-736–DOI: 10.1631/jzus.A1100029.

[4]Gribniak, V.;Cervenka, V.;Kaklauskas,G. (2013) Deflection prediction of reinforced concrete beams by design codes andcomputer simulation. Engineering Structures, v.56, p. 2175-2186 –DOI: 10.1016/j.engstruct.2013.08.045.

[5]Butean, C.;Heghes, B. (2020) Flexure Behavior of a Two Layer Reinforced Concrete Beam. Procedia Manufacturing, v. 46 p. 110-115 –DOI: 10.1016/j.promfg.2020.03.017.

[6]Bazant, Z. P. & Gambarova, P.G. (1984). Crack shear in concrete: Crack band microplane model. Journal of Structural Engineering. 110, 2015-2036.

[7]Bazant, Z. P. & Oh, B. H. (1985). Microplane model for progressive fracture of concrete and rock. Journal for Engineering Mechanics. 111, 559-582.

[8] LUBLINER, Jacob et al. A plastic-damage model for concrete. International Journal of solids and structures, v. 25, n. 3, p. 299-326, 1989.

[9] LEE, Jeeho; FENVES, Gregory L. Plastic-damage model for cyclic loading of concrete structures. Journal of engineering mechanics, v. 124, n. 8, p. 892-900, 1998.

[10]Simulia. Abaqus analysis user's manual. USA, 2012.

[11]Oliveira Filho, J., Alva, G. M. S., Canha, R. M. F., & El Debs, A. L. H. C. (2011). Comportamento cíclico de vigas de concreto armado: estudo experimental e análise numérica. IBRACON Structures and Materials Journal, 4(5), 729-733.

[12]Reginato, Luan et al. NLFEA of Reinforced Concrete Corbels: Proposed Framework, Sensibility Study, and Precision Level. Buildings, v. 13, n. 7, p. 1874, 2023.