Journal of Structural Engineering & Applied Mechanics (2024) 7(3):148-171
DOI 10.31462/jseam.2024.03148171

golden light publishing®

RESEARCH ARTICLE

Numerical evaluation of the performance of UHPC beams under bending and shear loads using different material models

Ayodele Akin-Adamu[1*], Çağlar Yalçınkaya[2], Onur Merter[3]

[1] FAMU-FSU College of Engineering, Department of Civil Engineering, Florida, USA
[2] Dokuz Eylül University, Department of Civil Engineering, İzmir, Türkiye
[3] İzmir University of Economics, Department of Civil Engineering, İzmir, Türkiye

Article History

Received 6 May 2024
Accepted 9 September 2024

Keywords

UHPC beams
Menetrey-Willam concrete model
Microplane concrete model
Drucker-Prager concrete model
Finite element analysis

Abstract

The Microplane and Menetrey-Willam concrete models are formulated on the research work of Bazant and Gambarova and Menetrey-Willam in the nineties respectively. They are used in this thesis to predict how reinforced Ultra-High-Performance Concrete (UHPC) beams will behave when subjected to flexural and shear dominant loadings. The numerical models are available in ANSYS, and they can simulate UHPC's extraordinary mechanical properties, such as high compression and tension strength, strain hardening, and softening behavior in compression and tension. In this research, four UHPC beams with dimensions of 180 wide by 270 deep by 4000 mm long with tensile reinforcements were studied using the two cementitious composite models mentioned above with the ANSYS software. The beams were investigated under flexural four-point loading and shear three-point loading with the ANSYS program. A quarter-geometric section model of the full beam was considered for the four-point loading to take advantage of symmetry in the longitudinal and transverse directions of the beam. For the three-point loading, the full geometry model of the beam was considered only because of the lack of symmetry. The results from the ANSYS finite element software were validated with an experimental study conducted by different researchers to show its capacity to predict the bending and shear behavior of reinforced UHPC beams. The comparisons of the results of both material models show that they can simulate the behavior of UHPC beams by using experimental load-deflection plots of the beam specimens and force strain plots of the longitudinal tensile reinforcements in the sample beams.

1. Introduction

The advent of digital computers and advancements in structural analysis over the past few decades have greatly influenced engineering. Argyris [3] highlights that the strong connection between structural theories and computers led to the development of large-scale finite element software, widely used across engineering disciplines. Among these, ANSYS stands out due to its extensive library of free online courses and its accessibility to students with minimal restrictions.

[*] Corresponding author (ayodele.akinadamu@ogr.deu.edu.tr)
eISSN 2630-5763 © 2023 Authors. Publishing services by golden light publishing®.
This is an open-access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

Ultrahigh-Performance Concrete (UHPC) has been utilized in the construction industry for the past 3–4 decades, particularly in developed countries like the USA, China, and France. Its use in developing and underdeveloped countries is limited due to a lack of technical knowledge and high production costs. UHPC is primarily employed in bridge construction, including decks, girders, piers, and foundations. The invention of UHPC stems from significant advancements in conventional concrete's material properties, achieved through precise material selection and the incorporation of fiber materials.

UHPC's high compressive strength (200–250 MPa) allows structures to be built with less material and weight, though its differing compression and tension behaviors from traditional concrete necessitate changes in structural design. El-Helou & Graybeal [10] investigated the flexural behavior of UHPC through extensive testing, including an 18.9 m span girder and a 4-point load setup. Their research led to the development of a flexural design framework. Reinforced UHPC beams typically exhibit localized crack failure due to their high compressive strength, post-cracking tensile strength, and short bond length of discrete fibers, which help bridge cracks before the rupture of tensile reinforcement. The fibers used were straight steel with dimensions of 13 mm in length, 0.2 mm in diameter, and a minimum tensile strength of 2,600 MPa.

Yoo et al. [25] investigated the bending of reinforced UHPC beams that failed due to compression crushing rather than typical crack localization, aiming to develop a design approach that fully leverages UHPC's high compressive strength. El-Helou & Graybeal [10] cautioned that ignoring UHPC's tensile strength can lead to oversimplified and unsafe predictions of failure modes and ductility in UHPC beams. To address this, structural finite element software that incorporates constitutive models like the Microplane and Menetrey-Willam models can be used.

Due to the high cost of UHPC, researchers have explored partially replacing the top section of traditional reinforced concrete beams with UHPC layers. This approach is believed to enhance flexural capacity and ductility in the compression zone of the beam. Dahish & Bakri [9] investigated this concept through experiments and finite element analysis using ABAQUS, concluding that the optimal UHPC layer thickness is approximately one-fifth of the beam's total height for a tensile reinforcement ratio of up to 3%.

Ahmad et al. [1] explored the strengthening of normal reinforced concrete beams with varying levels of damage by applying 30 mm thick UHPC layers to the tension region. The UHPC layers were placed on one, two, or three faces of the beams—bottom, side faces, and a combination of both. They compared the cracking load capacities of these strengthened beams to a control beam without UHPC. The study found that beams with UHPC on all three faces exhibited the highest cracking load capacity across all levels of pre-damage.

ANSYS structural finite element program provides different variations of 3D solid elements such as SOLID185, SOLID65, and SOLID186. SOLID185 is a three-dimensional brick element primarily used to model isotropic solid problems in finite element analysis. It consists of eight nodes, with each node possessing three translational degrees of freedom, corresponding to movements in the nodal *x*-, *y*-, and *z*-direction. SOLID186 is a three-dimensional, 20-node structural element used in finite element analysis. Each of its nodes has three translational degrees of freedom, corresponding to movements in the *x*-, *y*-, and *z*-directions. SOLID186 is the higher-order version of the SOLID185 element, providing enhanced accuracy for modeling complex geometries and material behaviors. SOLID65 is a specialized finite element used for modeling reinforced concrete structures or composite materials, such as fiberglass. This element is characterized by eight nodes, with each node featuring three translational degrees of freedom in the *x*-, *y*-, and *z*-directions. SOLID65 is particularly useful in analyzing both cracking under tensile stress and crushing under compressive stress [17].

During the loading of UHPC beams, microcracks form, leading to the degradation of the material's elastic stiffness and resulting in strain softening in both compression and tension properties. Bazant & Gambarova [5] explained that this degradation and the formation of microcracks violate Drucker’s stability postulate, a fundamental hypothesis of the theory of plasticity. They also noted that these microcracks complicate analysis as they cause the elastic stiffness to become anisotropic.

The Microplane model describes material behavior through stress-strain laws on multiple planes, effectively capturing damage orientations such as tensile cracking, friction, slip, compression splitting, and fiber reinforcement alignment. The model, available in ANSYS, is based on the research of Bazant & Byung [4] and Bazant & Gambarova [5]. It is particularly well-suited for engineering materials with diverse aggregate compositions, like concrete, where fine and coarse aggregates are embedded in a weak cement matrix. While the microplane model has been widely used to simulate concrete materials with finite element techniques, its application to UHPC materials remains limited.

The Menetrey-Willam model is a constitutive material model used to simulate the behavior of bonded aggregates like concrete, particularly under triaxial conditions. It is based on the Willam-Warnke yield surface and is employed for analyzing concrete failure in three-dimensional structures [16,23]. This research aims to develop a step-by-step procedure for capturing the flexural and shear behavior of UHPC beams at various stages: first cracking, reinforcement yielding, localized crack development, and compression failure, using the ANSYS finite element program. The procedure utilizes a combination of the concrete plasticity-damage microplane model and the Menetrey-Willam model to capture the elastic, plastic, and damage behavior of UHPC beams reinforced with conventional steel bars. ANSYS is used to define the 3D geometry of the beam models, characterize material properties, and perform finite element analysis based on 3D stress and strain concepts. The program replicates experimental results for load-deflection, compression stress-strain, and tension stress-strain curves, and generates color plots to visualize damage in the test beams.

2. Analytical research program

This section details the construction of numerical models in ANSYS to track the structural behavior of reinforced UHPC beams under shear and flexural loads. Structural analysis was conducted using a displacement control technique, incrementally increasing the beam's load until failure. The following subsections provide an in-depth description of the numerical techniques, including geometric modeling, discretization, and material modeling.

2.1. Geometries and schedule of test specimens

For the numerical analysis in this research, four reinforced rectangular UHPC beams were tested under displacement-controlled loading to failure in flexure and shear. The cross-sectional dimensions and reinforcement details of the beams are illustrated in Fig. 1. Beam sections U-B3 and U-B4 contain three 13 mm diameter straight steel bars in the bottom layer, while U-B5 and U-B6 have four 13 mm diameter bars. The 13 mm diameter reinforcement is grade 700 MPa, with a cross-sectional area of 132.73 mm² and a maximum ultimate strength of 700 MPa. The beams have a gross cross-sectional area of 4.86×10^4 mm² and a moment of inertia about the centroid (x-axis) of 2.95×10^8 mm⁴.

2.2. Test setup

Beams U-B3 and U-B5 were tested under 2-point loads applied on the top face of the beams at 432 mm on either side of the mid-span as shown in Fig. 2. The effective shear depth ratio of U-B3 and U-B5 is 5.95. This test setup is designed to simulate pure bending between points of load application. Beams U-B4 and U-B6 were subjected to a single point load applied on the top face of the beams at the distance of 610 mm from support (shear span) to examine the shear capacity as compared to the bending moment of the beams (see Fig. 3) [20]. The effective shear depth ratio of U-B4 and U-B6 is 2.6.

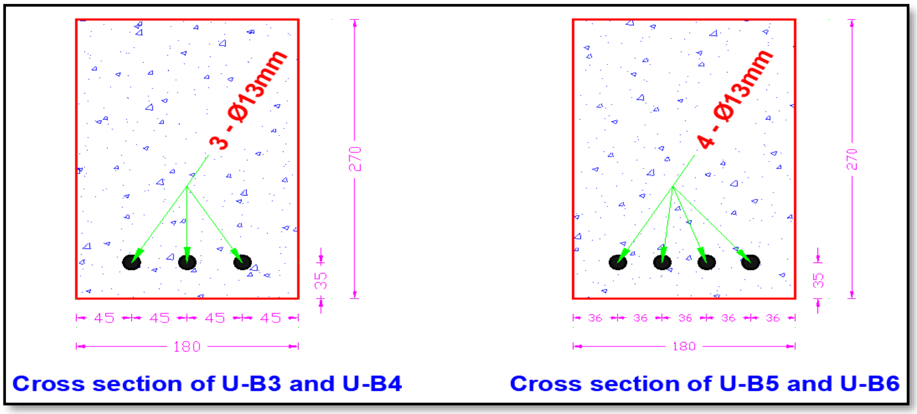


Fig. 1. Cross-sectional dimensions and reinforcement details of the tested beams, mm

2.3. Material characterization

The primary objective of this study is to predict the flexural and shear response of the experimentally tested beams using material models in ANSYS software. The mechanical properties were determined from uniaxial compression and tension tests, which provided data on UHPC properties such as density, Poisson's ratio, compression parameters, tension properties, and biaxial compression parameters at the time of testing. The average results from these tests are presented in Table 1.

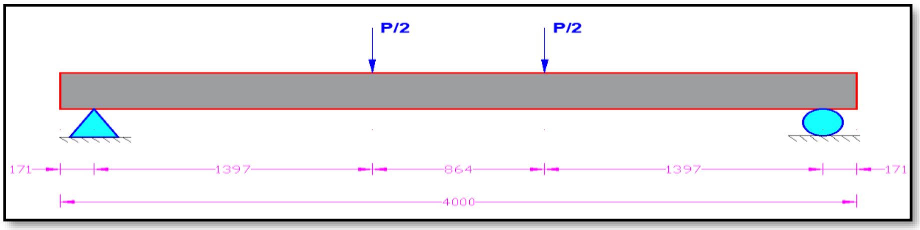


Fig. 2. Flexural loading (Beams U-B3 & U-B5), Units in mm

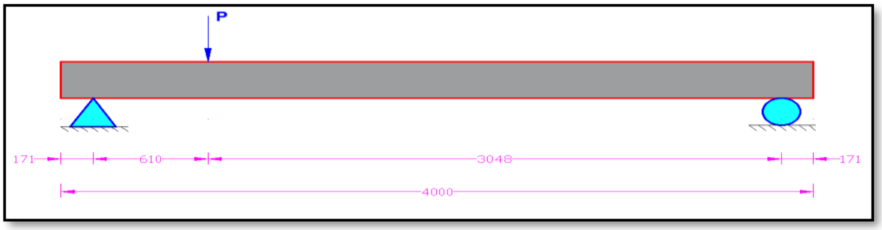
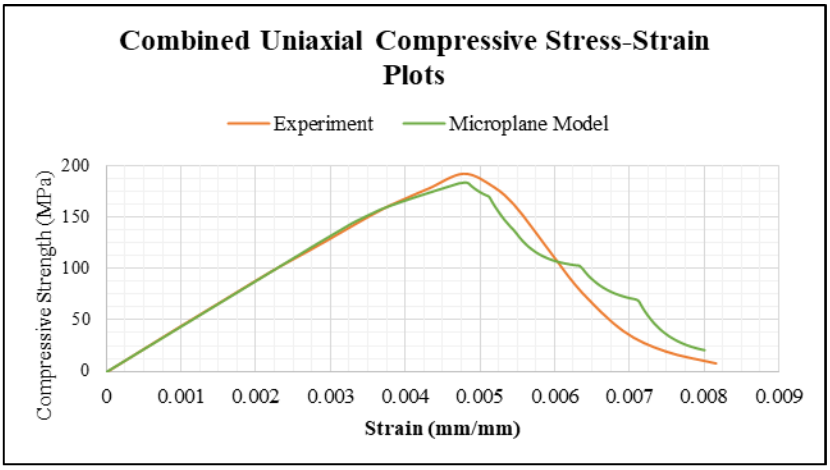


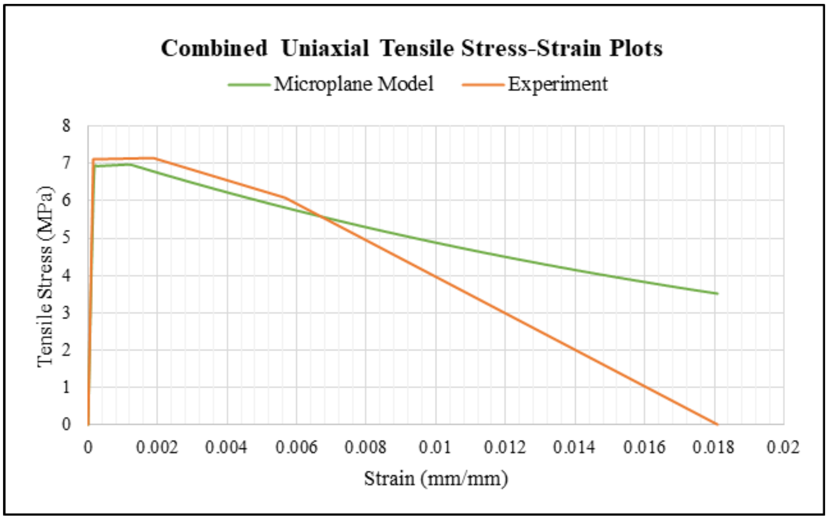
Fig. 3. Shear loading (beams U-B4 & U-B6), Units in mm

Table 1. Average density and uniaxial compressive and tensile parameters of the flexural and shear test

Material	Density, $\overline{w}_c$ (kg/m^3)	Poison ratio	Compression				Tension			Biaxial compression, $\bar{f}_b'$ (MPa)
			E_c×10^3 (MPa)	$\bar{f}_c'$ (MPa)	ε_p×10^{-3}	ε_{uc}	f_t (MPa)	ε_{cr}×10^{-4}	ε_{ut}×10^{-2}	
UHPC	2565	0.2	43.97	193	4.4	0.008	7.11	1.6	1.81	223.88



(a) Compressive loading



(b) Tensile loading

Fig. 4. Uniaxial stress-strain response for UHPC [20]

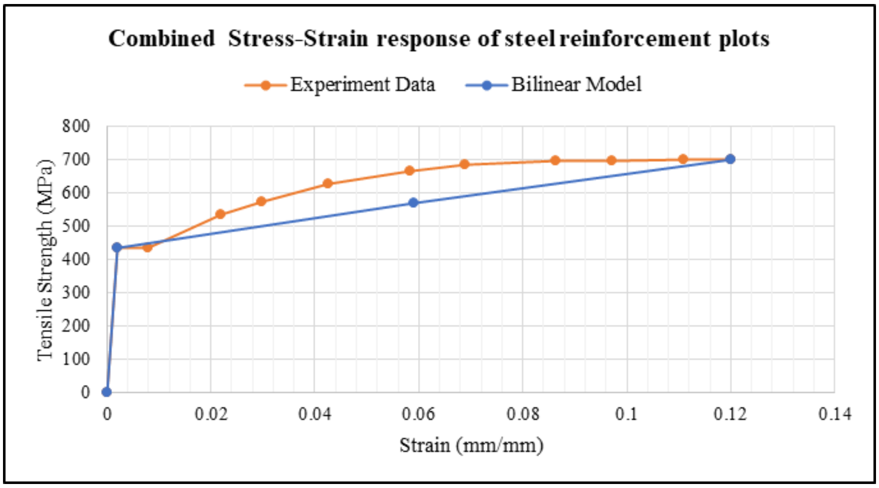


Fig. 5. Stress-strain response of steel reinforcement [20]

The experimental uniaxial compressive and tensile stress-strain curves for UHPC are shown in Figs. 4a and 4b and are compared with the microplane model results. The tensile longitudinal stress-strain curves for the steel reinforcements were also determined experimentally and are presented in Fig. 5, where a bilinear model was used to represent the behavior of the reinforcing steel.

The yield strength of the steel reinforcement was assessed at 0.21% strain. The steel test specimen had a density of 7850 kg/m³, a Poisson's ratio of 0.3, an average yield strength f_y of 435 MPa, an average ultimate tensile strength f_u of 700 MPa, and an average rupture strain ε_u of 0.12. The rebar's modulus of elasticity E_p was found to be 207 GPa. Strain data were recorded up to a uniaxial strain of approximately 0.12.

2.4. Material representation of UHPC in ANSYS

Two different material models are used to represent UHPC in the ANSYS software for this research. These include the following:

2.4.1. Coupled damage-plasticity microplane model

Using Mechanical APDL codes in ANSYS software, the following parameters were defined after performing uniaxial cyclic compression and tension tests. These parameters are illustrated in Fig. 6. Two 1×1×1 mm cubes were modeled in ANSYS and restrained in three orthogonal directions using the symmetry tool. As shown in Fig. 4a, a compressive displacement load of 0.008 mm, corresponding to the maximum compression strain, was applied to one cube, while a tensile displacement load of 0.0181 mm, corresponding to the maximum tensile strain, was applied to the other. The two cubes modeled in ANSYS are depicted in Fig. 7.

2.4.2. Menetrey-Willam model

In the ANSYS engineering data section, the following Menetrey-Willam parameters are defined with the following values (shown in Table 2) to represent the structural UHPC material used in this research.

```
! This means these commands will be carried out during pre-processing
/PREP7

! Step 1: Define elastic properties of material
E = 43970           ! Young's Modulus, MPa
nu = 0.2            ! Poisson's Ratio, unitless

! Step 2: Define microplane model properties
!  - Approximations and typical ranges per intuition... your concrete may be different!
fc = 193             ! Uniaxial compressive strength, MPa
fb = 223.88             ! Biaxial compressive strength, MPa         (estimate as 1.15*fc if unknown)
ft = 7.11              ! Tensile strength, MPa                     (estimate as 0.1*fc if unknown)
Rt = 1              ! Tension cap hardening constant, unitless  (typically = 1)
D = 0.5e4            ! Compressive hardening constant, MPa^2     (typically between 1e4 and 50e4)
sigma_cv = -149.2533      ! Int. of comp. cap and DP yield, MPa       (always negative, approximately -2/3*fb or lower)
Rc = 2              ! Compression cap ratio constant, unitless  (typically = 2)
gamma_t = 1e-3         ! Tensile damage threshhold, unitless       (typically = 0)
gamma_c = 15e-5      ! Compressive damage threshhold, unitless   (typically between 1e-5 to 10e-5)
beta_t = 50        ! Tension damage evolution, unitless        (typically 1.5*beta_c)
beta_c = 30000       ! Compression damage evolution, unitless    (typically between 1000 and 10000)

! Step 3: Define nonlocal parameters c and m
c = 10000             ! Nonlocal range parameter, mm^2            (element size should be < 0.5*sqrt(c))
m = 2               ! Over-nonlocal parameter, unitless         (typically 1 to 3)


! Step 4: Select element type
element = 215       ! Element type number (Linear = 215, Quadratic = 216)
```

Fig. 6. Coupled damage-plasticity micro-plane parameters

```
! Assign Elastic Properties
MP,EX,conc,E        ! Assign Young's Modulus
MP,NUXY,conc,nu     ! Assign Poisson's Ratio

! Assign microplane properties for coupled damaged-plasticity (MPLANE-DPC)
TB,MPLANE,conc,,,DPC
TBDATA,1,fc,fb,ft,Rt,D,sigma_cv
TBDATA,7,Rc,gamma_t,gamma_c,beta_t,beta_c

! Assign nonlocal properties to microplane model (MPLANE-NLOCAL)
TB,MPLANE,conc,,,NLOCAL
TBDATA,1,c,m

! Define coupled elements (this code assumes ITYPE = matid, which it should be in almost all cases)
itype = conc

et,itype,element
keyopt,itype,18,2      ! Element needs 2 nonlocal parameters for keyopt(18)

! Switch element type of those that are assigned Concrete
esel,s,mat,,conc
emodif,all,type,itype

! To check that everything worked out, print the element list
allsel
etlist,all

! Now proceed to the solution and output all results
/SOLU
```

Fig. 6. Continued

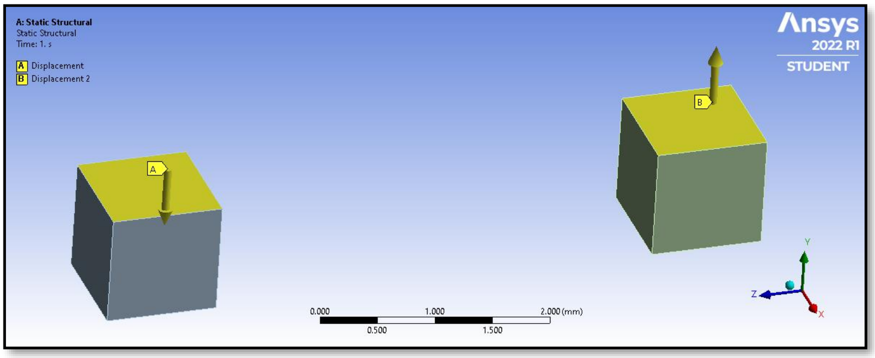


Fig. 7. Uniaxial cyclic compression and tension test cubes on ANSYS

Table 2. Menetrey-Willam UHPC parameters on ANSYS

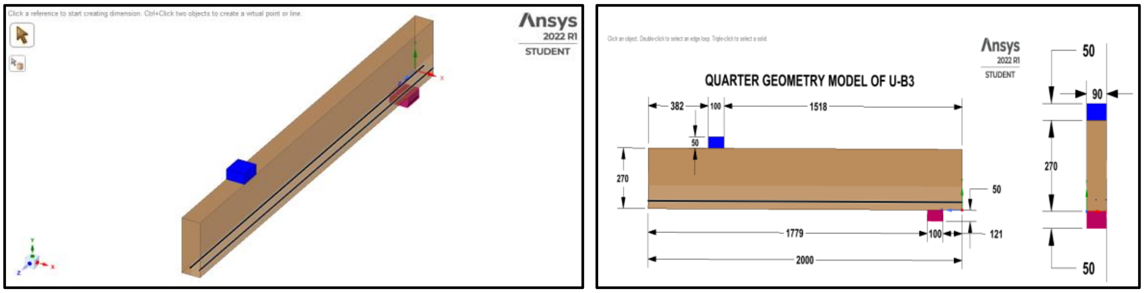
UHPC	
Structural	
Isotropic Elasticity	
Young's Modulus	43970 MPa
Poisson's Ratio	0.2
Bulk Modulus	24428 Mpa
Shear Modulus	18321 MPa
Menetrey-William	
Menetrey-William Base	
Uniaxial compressive Strength	193 MPa
Uniaxial Tensile Strength	7.11MPa
Biaxial Compressive Strength	223.88 MPa
Dilatancy Angle	30 degrees
Softening	

Table 2. Continued

Active Table	Linear
Plastic Strain at Uniaxial Compressive Strength	0.0044
Ultimate Effective Plastic Strain in Compression	0.008
Relative Stress at the Start of Nonlinear Hardening	0.95
Residual Compressive Relative Stress	0.08
Plastic Strain Limit in Tension	0.02
Residual Tensile Relative Stress	0.2

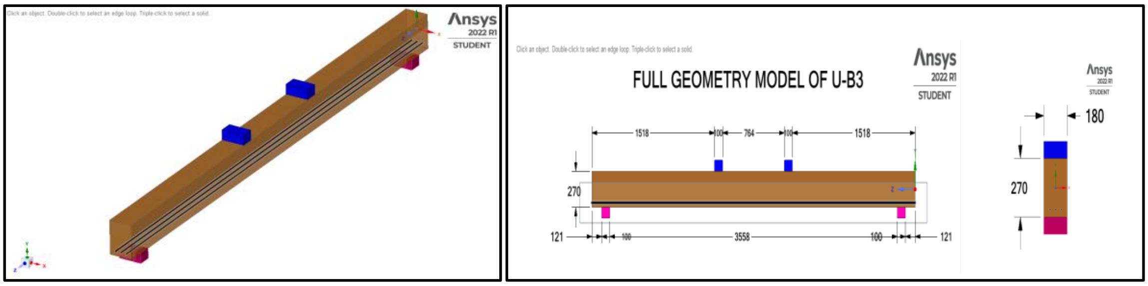
2.4.3. Geometry model of some of the reinforced UHPC beams on ANSYS

U-B3 contains three tensile reinforcements at the bottom and is subjected to a 4-point load system. Due to the beam's symmetric geometry and loading in two directions, a quarter model of the full geometry was used for numerical analysis, as shown in Fig. 8. Additionally, the full geometry model of U-B3 was also used for numerical analysis, as depicted in Fig. 9.



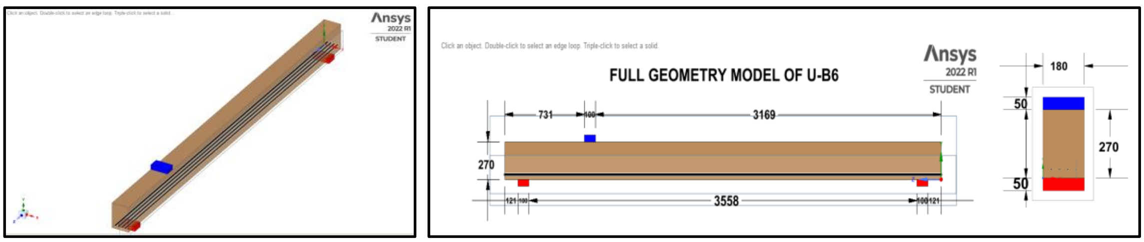
(a) 3D quarter geometry model of U-B3 (b) Quarter geometry model of U-B3 in mm

Fig. 8. U-B3 quarter geometric model with Ansys



(a) 3D full geometry model of U-B3 (b) Full geometry model of U-B3 in mm

Fig. 9. U-B3 full geometric model with Ansys



(a) 3D full geometry model of U-B6 (b) Full geometry model of U-B6 in mm

Fig. 10. U-B6 full geometric model with Ansys

As shown in Figs. 1 and 3, U-B4 and U-B6 contain three tensile reinforcements at the bottom and are subjected to a 3-point load system. Since U-B6 is not symmetric in its loading and geometry, the full geometry was used for numerical analysis, as illustrated in Fig. 10.

2.5. ANSYS finite element procedure for the analysis problem

Two finite element methods, each based on different material model types, are used for the structural analysis of the four reinforced UHPC beams discussed in this research.

These methods include:

1. Menetrey Willam method
2. Coupled Damage-Plasticity Microplane method.

The step-by-step procedure for the finite element methods is listed below:

1. The engineering material models are defined in the engineering data tab of the static structural solver in ANSYS. These definitions include models for tensile reinforcement steel, support plates, loading plates, and UHPC. If the coupled damage-plasticity Microplane method is used, a dummy material is initially defined for UHPC with arbitrary values for Young's modulus E and Poisson's ratio υ. The UHPC material is then accurately defined using the Mechanical APDL code.
2. The geometry model for the reinforced UHPC beams is defined under the geometry tab of the static structural solver in ANSYS. This tab opens a 3D modeling environment (SpaceClaim) where the geometry of the beams is created.
3. Material assignments, boundary condition assignments, and finite element discretization are specified in the model tab of the static structural solver in ANSYS. This tab opens the "Mechanical" window interface, where these assignments and discretizations are configured.
4. If the coupled damage-plasticity Microplane method is used, ANSYS Mechanical APDL parameters are employed to redefine the UHPC dummy material created in Step 1.
5. The solution is analyzed to display the results, including deformation, stress, strain, various chart plots, and damage color plot representations of the beams.

3. Results and discussion

In this section, the numerical results for all reinforced UHPC beams using ANSYS software are presented in the form of color plots, charts, or tables. The results for the numerical methods discussed in the previous section are reported for all test beams. For validation, the ANSYS findings are compared to experimental data. Key numerical results for the reinforced UHPC beams are summarized in Table 3. Data images from the reference paper were digitized to combine the experimental and finite element solutions for the sample beams.

3.1. Presentation of beam U-B5 ANSYS results with experimental data

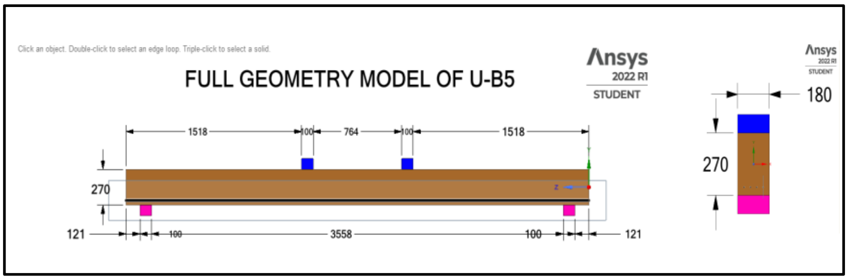
The midspan deflection at the bottom of the U-B5 beam is plotted against the total applied load (kN) on the two steel plates at the top of the beam in Fig. 11a. The U-B5 reinforced UHPC beam, with dimensions of 180×270×4000 mm, consists of four tensile reinforcements at the bottom section, as shown in Fig. 11(a). The beam was analyzed using ANSYS by three methods:

1. The full geometry section of U-B5 was analyzed using the Microplane material model.
2. A quarter geometry section was analyzed using the Microplane material model.
3. The quarter geometry section was analyzed using the Menetrey-Willam model.

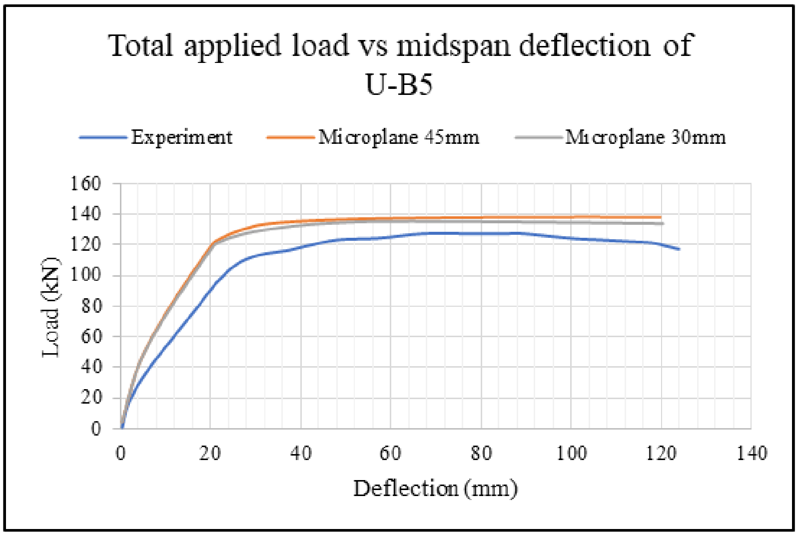
Additionally, mesh sensitivity analysis was conducted for U-B5 using 45 mm and 30 mm mesh sizes with the Microplane material model. Due to high computational costs, only these two mesh sizes were considered. The beam was discretized using a 45 mm and 30 mm hexagonal finite element mesh in ANSYS. For both the full and quarter geometry models, the beam was tested using a 4-point load test to capture its flexural behavior. An equal displacement loading of 105 mm was applied to the two steel plates at the top of the U-B5 beam. To simulate pinned support, a displacement constraint of 0 mm in the *x*, *y*, and *z* directions was applied under one of the support steel plates. For the other support plate, a displacement constraint of "free, 0 mm, and free" in the *x*, *y*, and *z* directions, respectively, was applied to simulate roller support.

Fig. 11b presents three comparison plots. One of the plots is from the research by Solhmirzaei & Kodur [20], which recorded a maximum applied load of 126.6 kN and a midspan deflection of 120 mm. The other two plots show the results from the full geometry structural analysis in ANSYS using the microplane model with 45 mm and 30 mm linear 3D finite element meshes. Under a 105 mm displacement loading at the top of the beam, the 45 mm mesh analysis estimated a maximum applied load of 138.24 kN and a midspan deflection of 119.62 mm, while the 30 mm mesh analysis estimated 135.07 kN and 120.39 mm, respectively.

The load-deflection curves from ANSYS closely resemble the experimental plot, although the microplane model in ANSYS overestimates the load capacity of U-B5 by about 6.7%, and the model's response is more rigid than observed in the experiment. This discrepancy can be attributed to the presence of microcracks and imperfections in the beam specimen, which are not accounted for in the model. Nonetheless, the results indicate that the Microplane model in ANSYS is a viable tool for predicting the flexural behavior of reinforced UHPC beams.



(a) Full geometric U-B5



(b) Combined plot of Microplane modal and experimental data

Fig. 11. Total applied load vs midspan deflection of U-B5

Table 3. Tabular presentation of key thesis results [20]

				ANSYS FE results											
	Experimental results			Microplane Material model									Menetrey-William model		
				Full Geometry Model						Quarter Geometry Model					
Beam label	Load capacity (kN)	Max midspan deflection (mm)	Total strain in longitudinal reinforcement (mm/mm)	Load capacity (kN)	Ratio	Max deflection (mm)	Ratio	Total strain in longitudinal reinforcement (mm/mm)	Max rebar tensile Stress (MPa)	Load capacity (kN)	Max deflection (mm)	Max rebar tensile Stress (MPa)	Load capacity (kN)	Max deflection (mm)	Max rebar tensile Stress (MPa)
	(1)	(2)		(3)	(1)/(3)	(4)	(2)/(4)								
U-B3	97.1	a	0.025	116.27	0.835	121.72	a	0.035311	509.57	120.33	122.24	511.14	98.512	132.19	635.33
U-B4	142.1	a	a	151.64	0.937	89.999	a	0.050354	543.47	b	b	b	b	b	b
U-B5	126.6	122	0.025	135.07	0.937	120.39	1.01	0.033968	506.64	140.15	121.15	508.32	115.82	121.73	606.36
U-B6	177.1	83	a	176.4	1	88.71	0.936	0.049332	541.17	b	b	b	b	b	b

[a]Data not reported

[b]ANSYS results are not available because the loading is not symmetric on U-B4 and U-B6

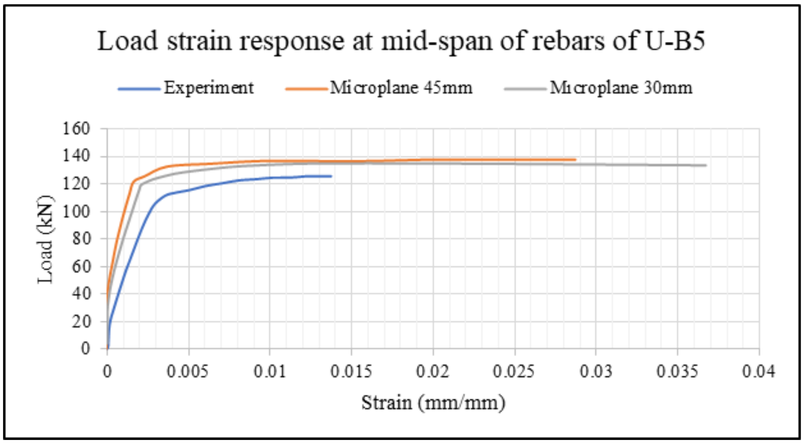
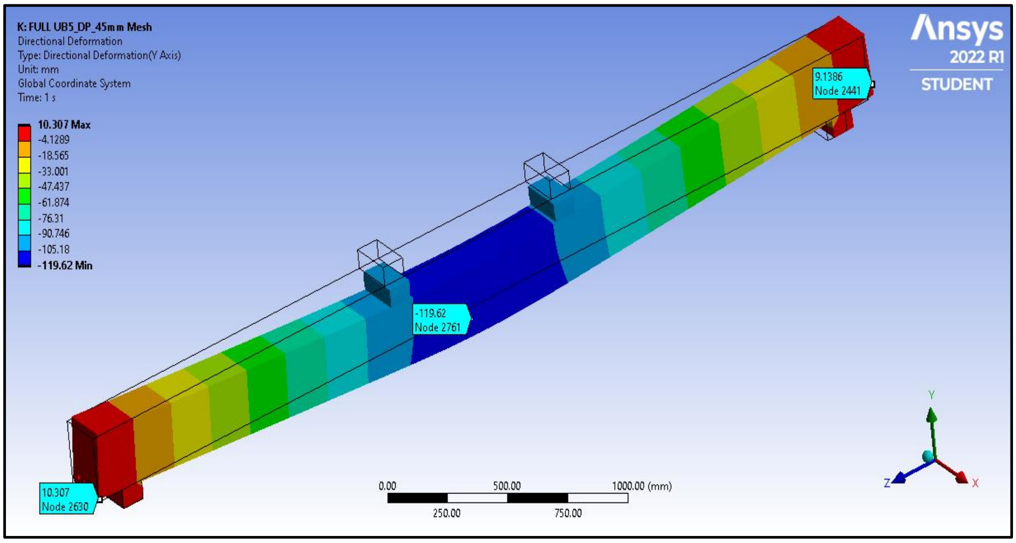
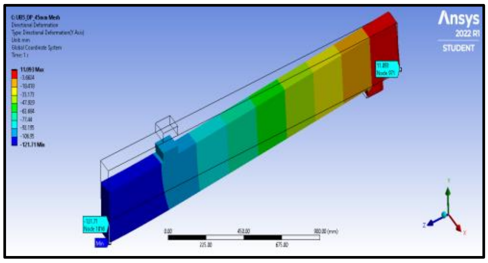


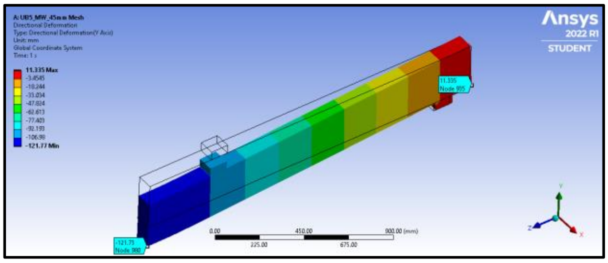
Fig. 12. Load strain response at mid-span of rebars of U-B5, combined experimental plot and ANSYS microplane model plot



(a) Full geometry microplane model plot



(b) Quarter geometry microplane model plot



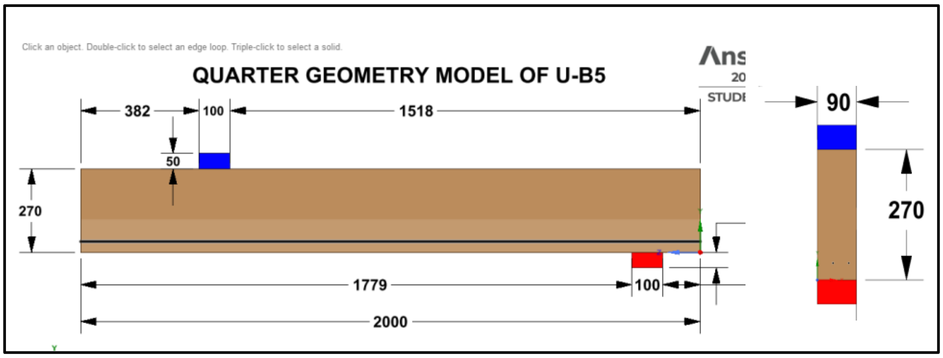
(c) Quarter geometry Menetrey Willam model plot

Fig. 13. U-B5 deflection color plot at failure load

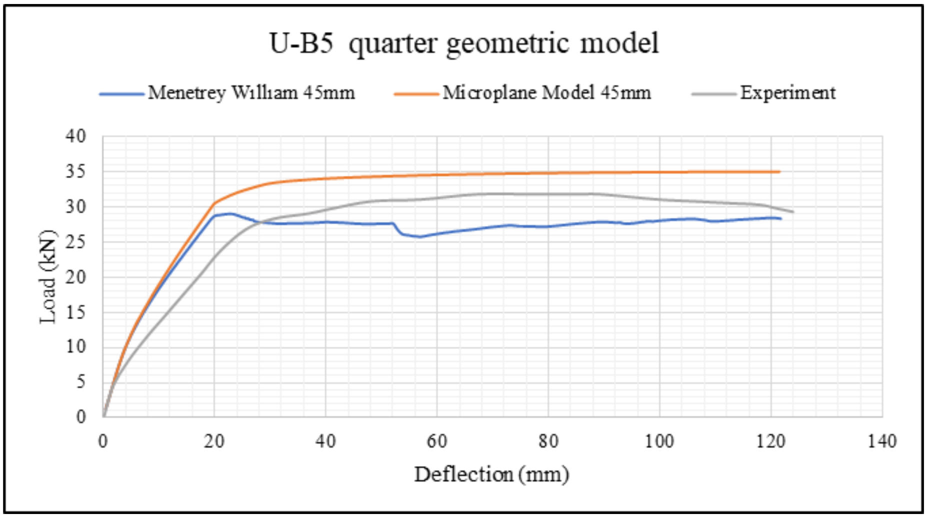
Fig. 12 presents a combined experimental and microplane model plot of the total applied load (kN) on the two steel plates at the top of the U-B5 beam versus the total strain in the tensile reinforcement at the bottom section of the beam. The results for both 45 mm and 30 mm finite element meshes are also shown in

Fig. 12. Under a 105 mm displacement loading at the top of the beam, the microplane model with a 45 mm mesh estimated a total strain of 0.028848, while the 30 mm mesh calculated a total strain of 0.036671. The results in Fig. 12 indicate that the response of the rebars in this material model is stiffer than the experimental outcome. This discrepancy is due to the use of a bilinear tensile stress-strain model to approximate the hardening behavior of the steel rebars, which differs slightly from the actual mechanical properties of the reinforcement steel. Overall, the ANSYS plot in Fig. 12 closely matches the experimental plot.

Fig. 13 presents three color plots of U-B5 results obtained from full and quarter geometry structural analysis in ANSYS. Fig. 13a shows the color plot using the full geometry model with the Microplane material model, estimating a maximum midspan deflection of 119.62 mm. Fig. 13b depicts a quarter geometry model analysis using the Microplane material model, predicting a maximum midspan deflection of 121.71 mm. Fig. 13c, another quarter geometry model analysis using the Menetrey-Willam model, predicts a maximum midspan deflection of 121.73 mm. The experimental midspan deflection is 122 mm. The close agreement between the deflection results from these color plots and the experimental data indicates that both the Microplane and Menetrey-Willam material models are effective in predicting the flexural behavior of reinforced UHPC beams.



(a) U-B5, Units in mm

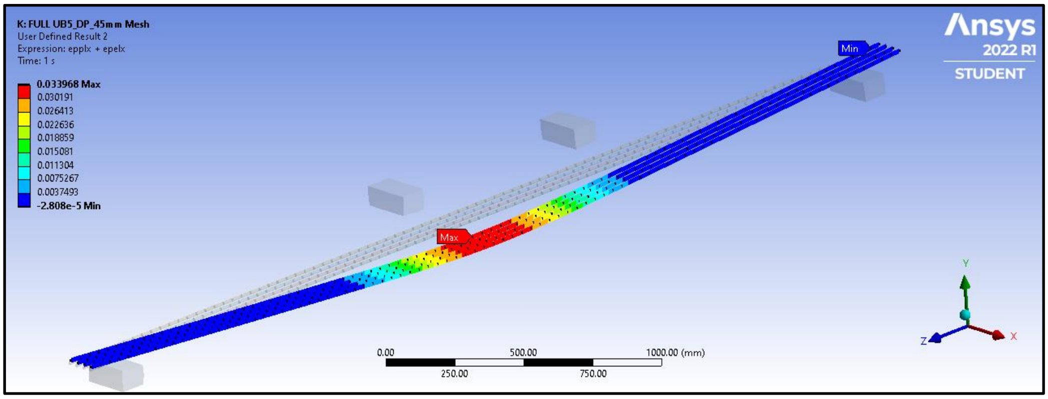


(b) Combined plot of Microplane model and experimental data

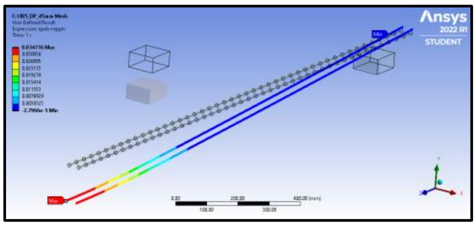
Fig. 14. Total applied load vs. midspan deflection of U-B5, quarter geometric ANSYS Microplane and Menetrey-Willam model plot

Fig. 14 presents combined plots of the applied load (kN) at the top of the U-B5 beam vs. the midspan deflection at the bottom using a quarter geometry model. The plots compare the Microplane and Menetrey-Willam models against experimental data, with a focus on the 45 mm mesh size. Under a 105 mm displacement loading at the top of the beam, the Microplane model estimates a maximum applied load and midspan deflection of 35.038 kN and 121.51 mm, respectively. For the Menetrey-Willam model, the total applied load on the quarter geometry beam model is calculated as 35.038×4 = 140.152 kN. Additionally, under the same 105 mm displacement, the Menetrey-Willam model estimates a maximum applied load and midspan deflection of 28.954 kN and 121.77 mm, with the total applied load on the beam calculated as 28.954×4 = 115.816 kN. The load-deflection curve from ANSYS for the Microplane model closely resembles the experimental plot, although the Microplane model overestimates the load capacity of U-B5, while the Menetrey-Willam model underestimates it. A correction factor could be applied to adjust for microcracks and imperfections in the actual beam. Overall, these plots demonstrate that both the Microplane and Menetrey-Willam material models in ANSYS can effectively predict the flexural behavior of reinforced UHPC beams. The difference in post-yield behavior between the two models may be due to the assumptions made in the Menetrey-Willam and Microplane models during the calibration of the stress-strain curves for the tension and compression behavior of UHPC.

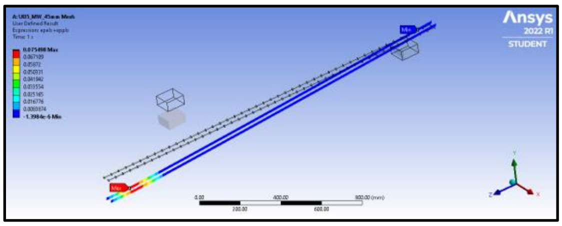
Fig. 15 presents the total strain color plots for the tensile reinforcements in the beam section, using the three methods mentioned earlier. The results discussed are for the 45 mm mesh. The maximum total strain predicted is 0.033968 mm/mm for the full geometry (Microplane), 0.034716 mm/mm for the quarter geometry (Microplane), and 0.075498 mm/mm for the quarter geometry (Menetrey-Willam) models. The Microplane material model plots, shown in Figs. 15(a) and (b), indicate that at least 30% of the length of the tensile reinforcements at the bottom section yielded. In contrast, only 10% of the reinforcement length yielded in Fig. 15(c) when using the Menetrey-Willam model.



(a) Full geometry microplane model plot

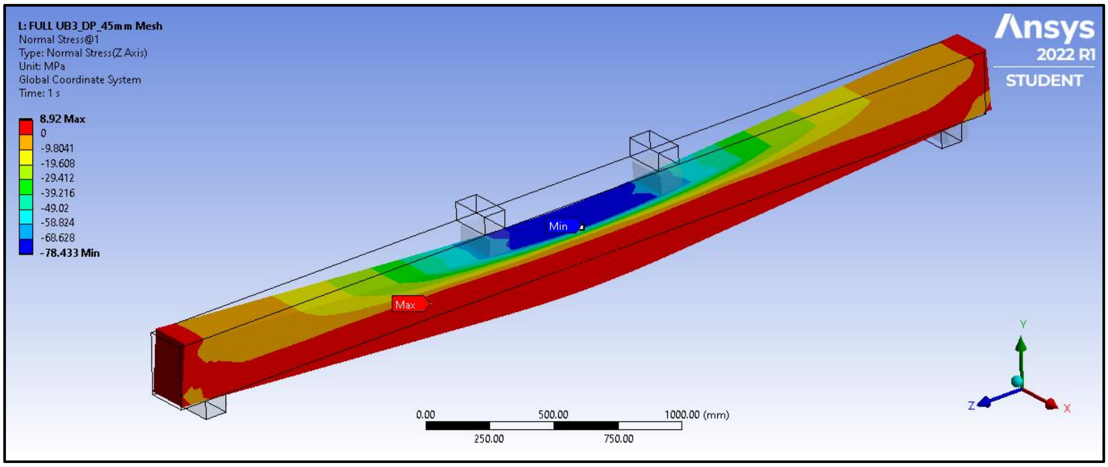


(b) Quarter geometry microplane model plot

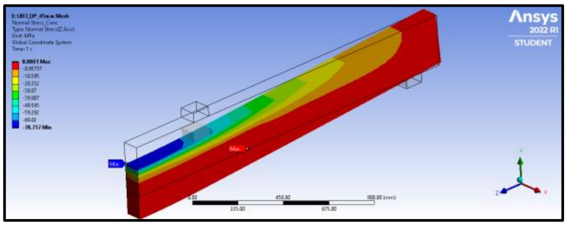


(c) Quarter geometry Menetrey Willam model plot

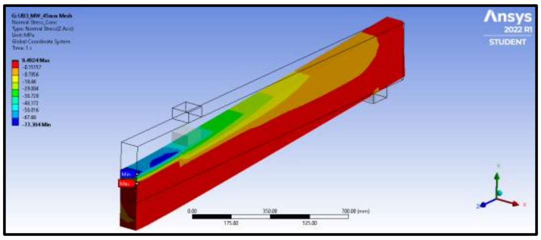
Fig. 15. Total strain color plot for tensile reinforcements at failure load in mm/mm for U-B5



(a) Full geometric ANSYS microplane model



(b) Quarter geometric ANSYS microplane model



(c) Quarter geometric ANSYS Menetrey Willam model

Fig. 16. Equivalent von Mises stress color plot for tensile reinforcements at failure load in MPa

Fig. 16 presents the equivalent von Mises stress color plots for the tensile reinforcements in the beam section, using the three methods mentioned earlier. The plots predict a maximum total stress of 506.64 MPa for the full geometry (Microplane), 508.32 MPa for the quarter geometry (Microplane), and 606.36 MPa for the Menetrey-Willam model.

3.2. Presentation of beam U-B3 ANSYS results with experimental data

Fig. 17 presents a combined plot of the total applied load (kN) on the two steel plates at the top of the U-B3 beam versus the midspan deflection at the bottom, using 45 mm and 30 mm finite element mesh sizes. The reinforced UHPC beam U-B3, with dimensions of 180×270×4000 mm, contains three tensile reinforcements at the bottom section, as shown in Fig. 17. The beam was analyzed in ANSYS using three methods:

1. The full geometry section of U-B3 was analyzed using the Microplane material model.
2. A quarter geometry section was analyzed using the Microplane material model.
3. The quarter geometry section was analyzed using the Menetrey-Willam model.

The plots in Fig. 17 display the results of the full geometry structural analysis using the Microplane model. Under a 105 mm displacement loading at the top of the beam, the 45 mm finite element mesh size results estimate a maximum applied load and midspan deflection of 119.02 kN and 120.59 mm, respectively. For the 30 mm finite element mesh size, the estimates are 116.27 kN and 121.72 mm, respectively.

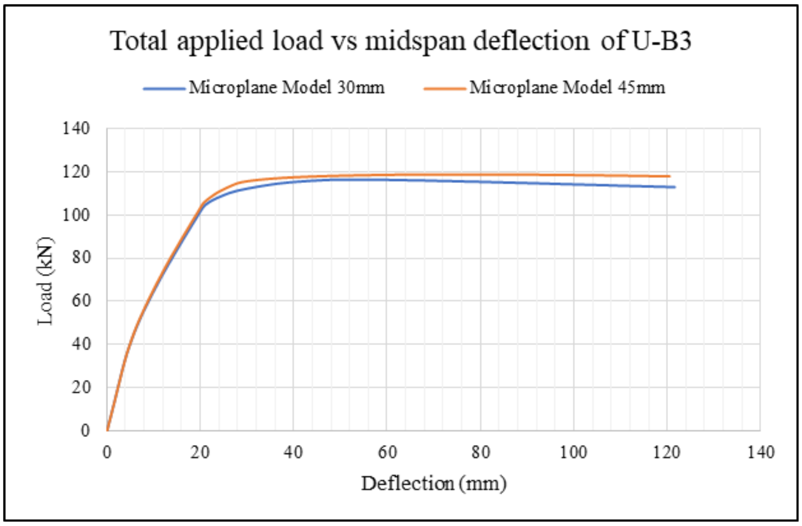


Fig. 17. Total applied load vs midspan deflection of U-B3

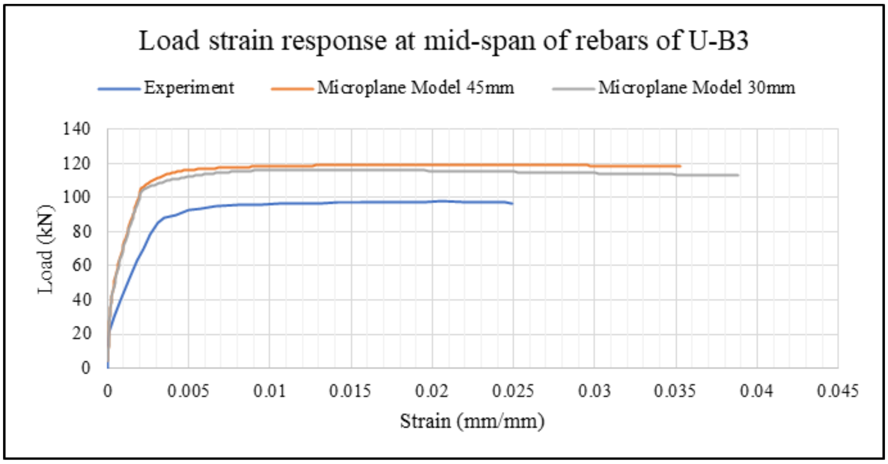


Fig. 18. Load strain response at mid-span of rebars of U-B3 combined experimental and ANSYS microplane model

Fig. 18 presents a combined plot of the total applied load (kN) on the two plates at the top of the U-B3 beam versus the total strain in the tensile reinforcement at the bottom section of the beam. The plot includes results from the research by Solhmirzaei & Kodur [20] and the finite element solution using the Microplane material model. The Microplane model was analyzed with 45 mm and 30 mm mesh sizes, recording a maximum total strain of 0.035273 and 0.038842, respectively. In comparison, the experimental data plot estimates a total strain of 0.025 under a 105 mm displacement loading at the top of the beam. As discussed in previous sections, the difference in results may be attributed to the presence of microcracks and imperfections in the beam specimen and the assumptions made during the calibration of the Microplane material model parameters. Despite these differences, the shape of the finite element plot from the Microplane model is compatible with the experimental data.

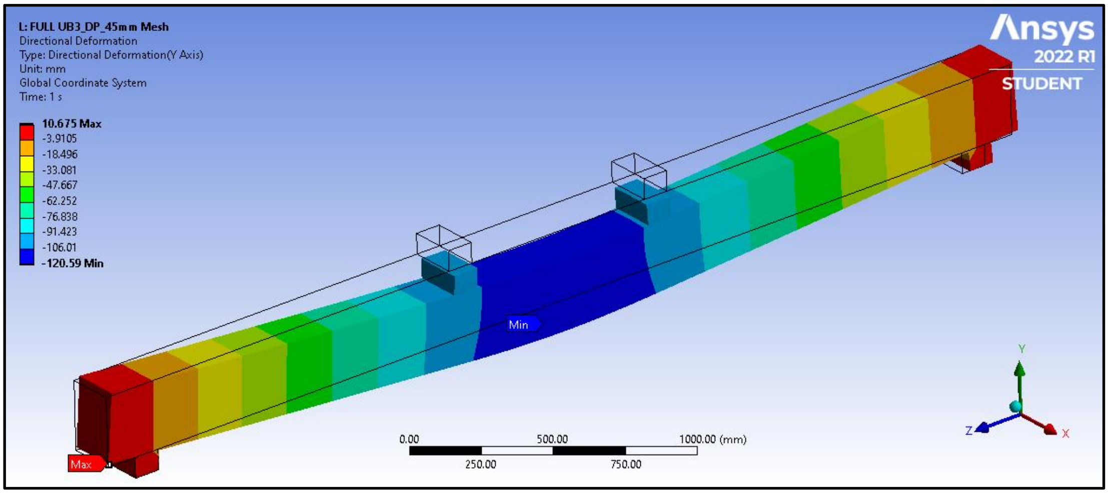
Fig. 19 presents three color plots of U-B3 results obtained using full and quarter geometry structural analysis in ANSYS with a 45 mm finite element mesh size.

- Fig. 19(a) shows the color plot using the full geometry model with the Microplane material model, estimating a maximum midspan deflection of 120.59 mm.

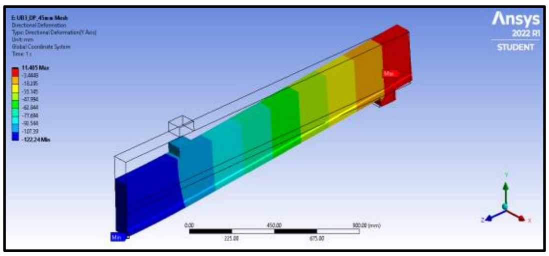
- Fig. 19(b) depicts a quarter geometry model analysis using the Microplane material model, predicting a maximum midspan deflection of 122.24 mm.
- Fig. 19(c) is a quarter geometry model analysis using the Menetrey-Willam model, predicting a maximum midspan deflection of 132.19 mm.

The experimental midspan deflection is 122 mm. The deflection estimates from the two material models vary by about 15%.

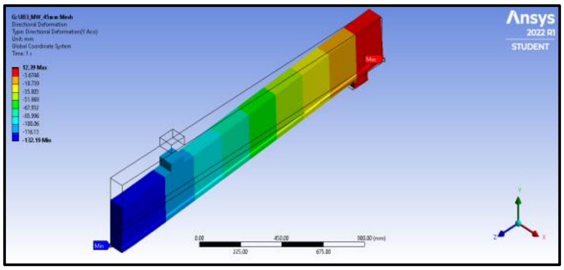
Fig. 20 presents plots of the applied load (kN) at the top of the U-B3 beam versus the midspan deflection at the bottom, comparing the Microplane and Menetrey-Willam material models using a quarter geometry model. The results discussed are for the 45 mm finite element mesh size. Under a 105 mm displacement loading at the top of the beam, the Microplane model estimates a maximum applied load and midspan deflection of 30.083 kN and 122.24 mm, respectively, with the total applied load on the beam estimated as 30.083×4 = 120.332 kN. For the Menetrey-Willam model, the maximum applied load and midspan deflection are estimated as 24.628 kN and 132.19 mm, respectively, with the total applied load on the beam calculated as 24.628×4 = 98.512 kN. The load-deflection curves from ANSYS in Fig. 20(b) closely resemble the experimental results, though the Menetrey-Willam model shows irregularities after the yielding point of the beam, whereas the Microplane model provides a smoother curve. This suggests that the Microplane model performs better than the Menetrey-Willam model in this context. Overall, both models in ANSYS can be used to predict the flexural behavior of reinforced UHPC beams.



(a) Full geometry microplane model plot

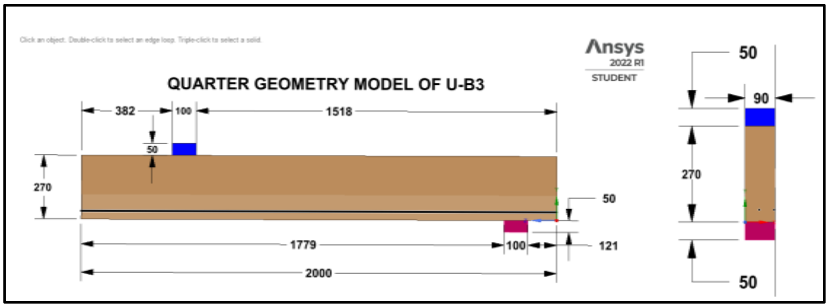


(b) Quarter geometry microplane model plot

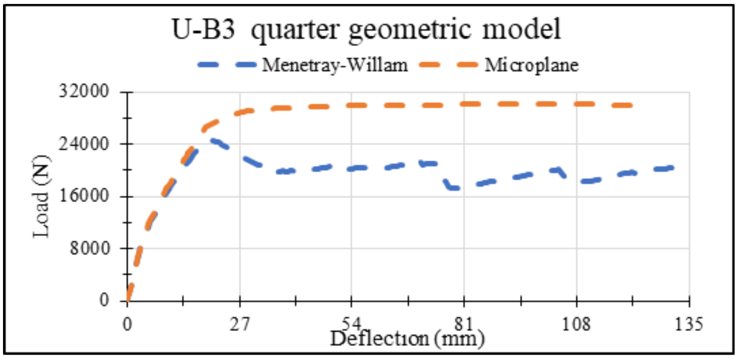


(c) Quarter geometry Menetrey Willam model plot

Fig. 19. U-B3 deflection color plot at failure load

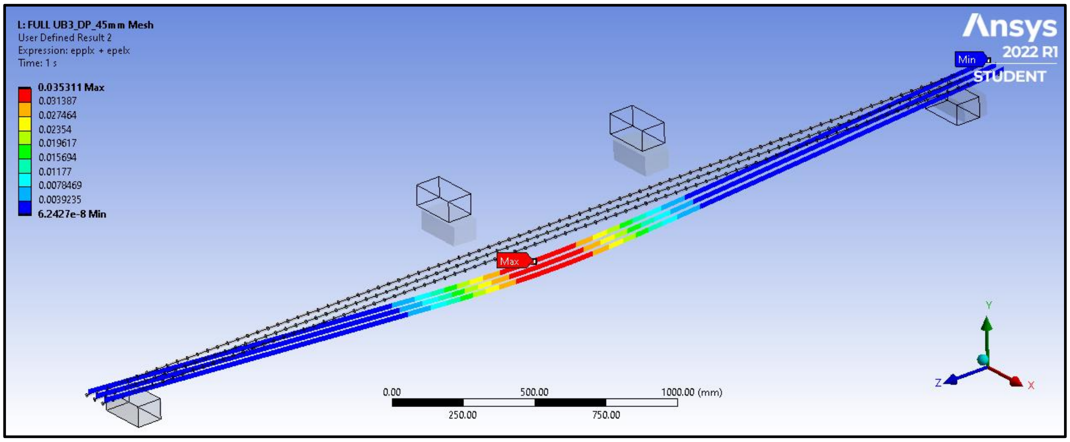


(a) Quarter geometry on ANSYS, mm

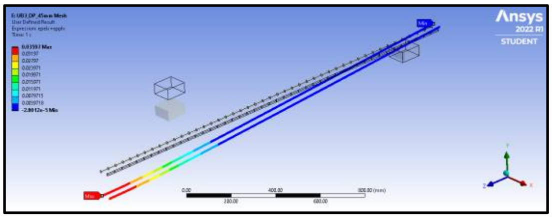


(b) ANSYS Microplane and Menetrey Willam model plot

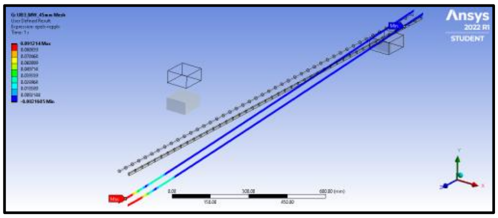
Fig. 20. Total applied load vs midspan deflection of U-B3



(a) Full geometry microplane model plot

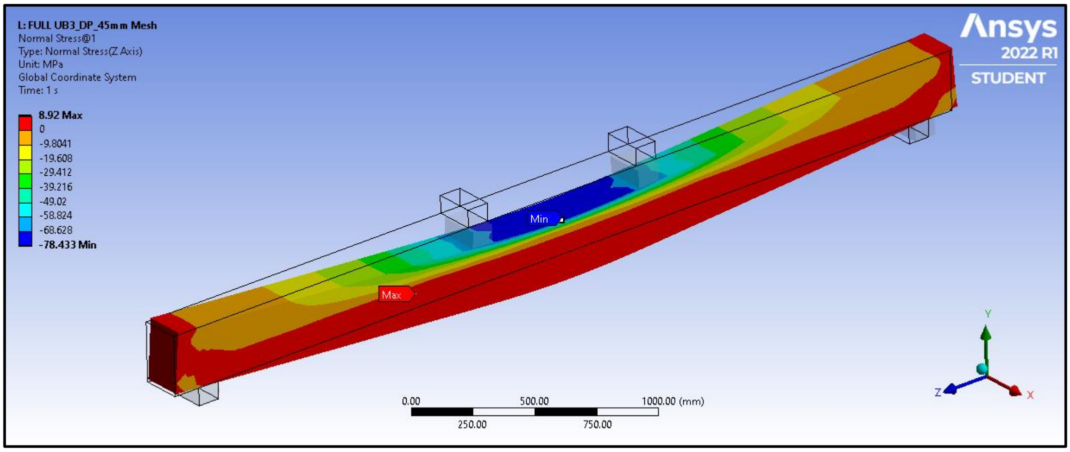


(b) Quarter geometry microplane model plot

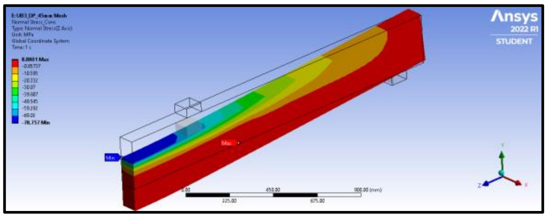


(c) Quarter geometry Menetrey Willam model plot

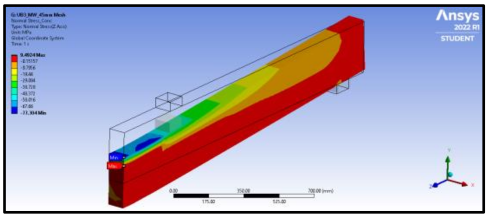
Fig. 21. Total strain color plot for tensile reinforcements at failure load in mm/mm for U-B3



(a) Full geometry microplane model plot



(b) Quarter geometry microplane model plot



(c) Quarter geometry Menetrey Willam model plot

Fig. 22. Normal stress color plot for U-B3 at failure load in MPa

Fig. 21 presents the total strain color plots for the tensile reinforcements in the beam section, using the 45 mm finite element results for the three methods mentioned earlier. The plots predict a maximum total strain of 0.035311 mm/mm for the full geometry (Microplane), 0.03597 mm/mm for the quarter geometry (Microplane), and 0.091214 mm/mm for the quarter geometry (Menetrey-Willam) models. The Microplane material model plots in Figs. 21(a) and (b) show that at least 35% of the length of the tensile reinforcements at the bottom section yielded, while only 12% of the reinforcement length yielded in Fig. 21(c) using the Menetrey-Willam model.

Fig. 22 presents the normal stress color plots for the beam section at failure load, using the 45 mm finite element results for the three methods mentioned earlier. Figs. 22(a) and 22(b) show identical results, with a maximum tensile stress of 8.9 MPa and a maximum compressive stress of 78.8 MPa. In contrast, the Menetrey-Willam material model, shown in Fig. 22(c), estimates a maximum tensile stress of 9.4924 MPa and a maximum compressive stress of 77.304 MPa.

3.3. Presentation of beam U-B6 ANSYS results with experimental data

Fig. 23 presents a combined plot of the total applied load (kN) on the two steel plates at the top of the U-B6 beam versus the midspan deflection at the bottom of the beam. The plot includes results from the Microplane material model and experimental data. The U-B6 beam, with dimensions of 180×270×4000 mm, consists of four tensile reinforcements at the bottom section. The Microplane material model was used to analyze the full geometry of U-B6, which was tested in ANSYS using 30 mm and 45 mm hexagonal finite element meshes. The beam was subjected to a 3-point load test to evaluate its shear behavior, with a 72 mm displacement loading applied at the top of the beam. For the support conditions, a displacement of 0 mm in the *x*, *y*, and *z* directions was applied to one of the steel plates to simulate pinned support, while the other plate had a displacement support of "Free, 0 mm, and Free" in the *x*, *y*, and *z* directions, simulating roller support. The experimental plot in Fig. 23 recorded a maximum applied load and midspan deflection of 177 kN and 83 mm, respectively. In comparison, the Microplane material model estimated a maximum applied load and deflection of 181 kN and 88.81 mm for the 45 mm mesh, and 176.4 kN and 88.71 mm for the 30 mm mesh. The load capacity of the beam decreased slightly from 181 kN to 176 kN when moving from the 45 mm to the 30 mm mesh size. Overall, the load-deflection curves from ANSYS closely resemble the experimental plot, indicating that the Microplane model in ANSYS is effective in predicting the shear behavior of reinforced UHPC beams.

Fig. 24 presents combined plots of the total applied load (kN) on the two plates at the top of the U-B6 beam versus the total strain in the tensile reinforcement at the bottom section of the beam, using 30 mm and 45 mm finite element meshes. These plots are the results of the full geometry structural analysis in ANSYS using the Microplane model. Under a 72 mm displacement loading at the top of the beam, the estimated total strain is 0.049332 for the 45 mm mesh and 0.0527 for the 30 mm mesh at the end of loading. However, this result could not be validated due to the absence of corresponding experimental data.

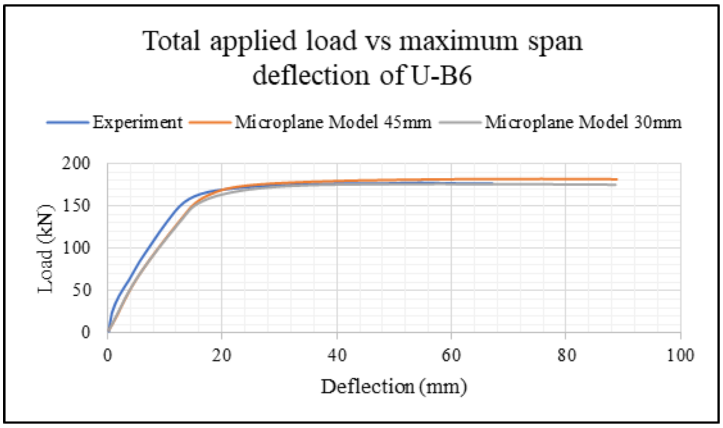


Fig. 23. Total applied load vs midspan deflection of U-B6 (Full geometry)

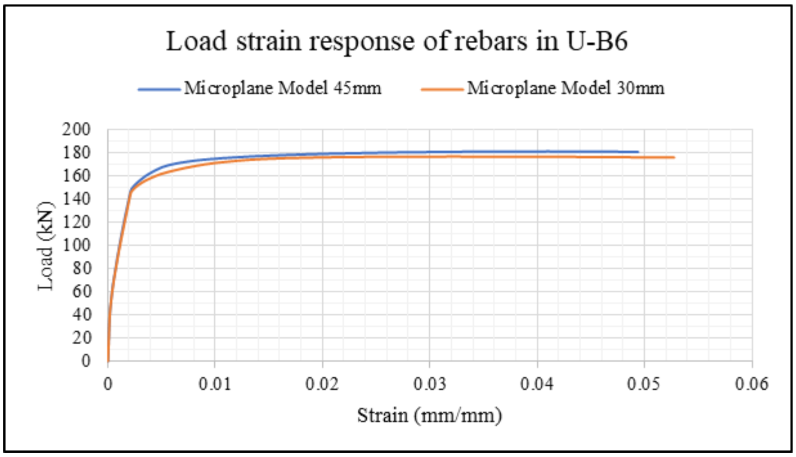


Fig. 24. Load strain response at mid-span of rebars of U-B5, ANSYS microplane model plot

Fig. 25 shows a load-deflection color plot of the U-B6 beam, obtained from a full geometry structural analysis in ANSYS using the Microplane material model. The result, based on a 45 mm finite element mesh, estimates a maximum midspan deflection of 88.81 mm.

Fig. 26 presents the total strain color plot from a full geometry structural analysis in ANSYS for the tensile reinforcements in the beam section. The analysis predicts a maximum total strain of 0.049332 mm/mm. The Microplane material model plot indicates that at least 20% of the length of the tensile reinforcements at the bottom section yielded.

Fig. 27 presents the normal stress color plot for the full geometry Microplane ANSYS analysis of the beam section at failure load. The analysis estimates a maximum tensile stress of 10.029 MPa and a maximum compressive stress of 98.785 MPa.

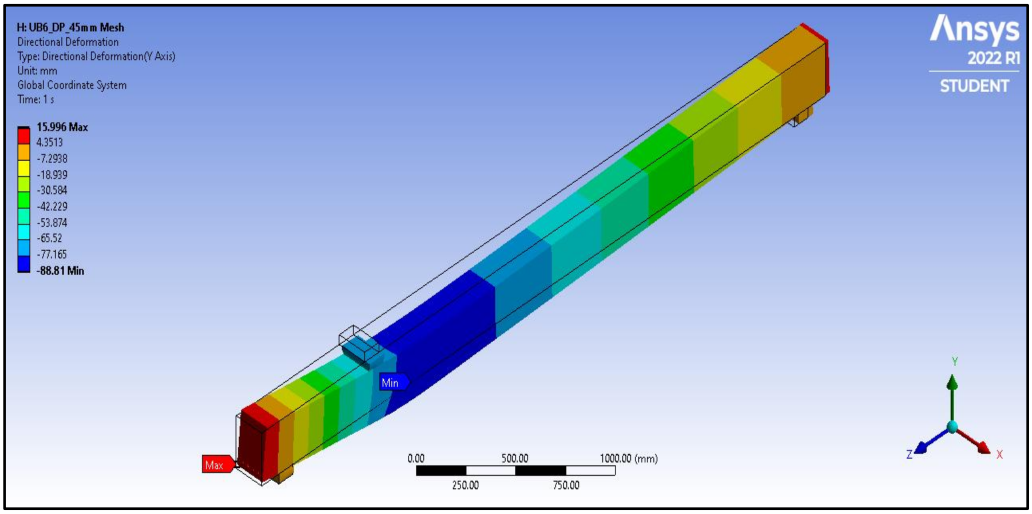


Fig. 25. U-B6 deflection color plot at failure load, full geometric microplane model plot

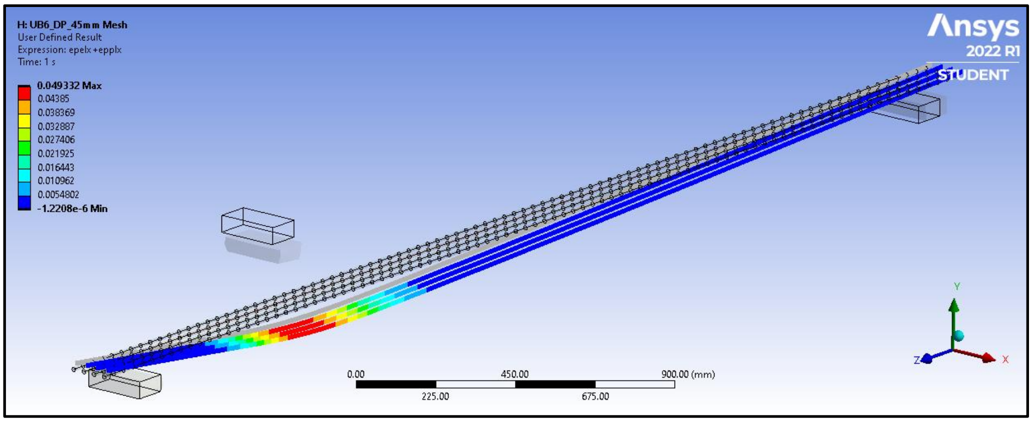


Fig. 26. Total strain color plot for tensile reinforcements at failure load in mm/mm, full geometric ANSYS microplane model

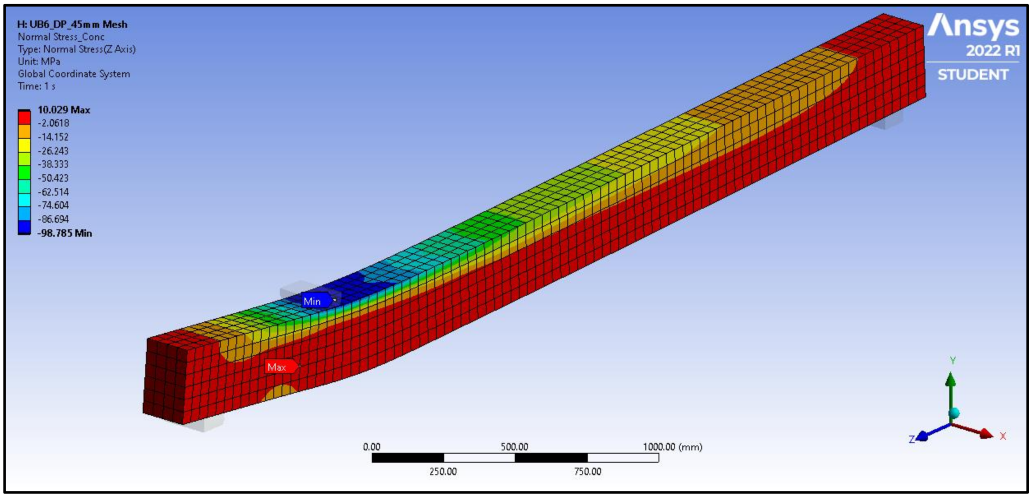


Fig. 27. Normal stress color plot for U-B6 at failure load in MPa, full geometric ANSYS microplane model

4. Conclusions

In conclusion, this study effectively employed the Microplane and Menetrey Willam concrete models within the ANSYS software to simulate the behavior of reinforced Ultra-High Performance Concrete (UHPC) beams under flexural and shear loading conditions.

The general conclusions obtainable from this research are highlighted below:

1. ANSYS finite element software can be used to analyze and design reinforced UHPC beams accurately when the concrete damage plasticity Microplane material model is used with the software.
2. It is possible to capture the behavior of reinforced UHPC beams under flexural and shear load when the concrete damage plasticity Microplane material model is applied.
3. It is possible to determine the extent of yielding in tension reinforcements provided in UHPC beams utilizing the finite element analysis.
4. Since the load midspan deflection curve and load strain response of tension rebars from the experiment were accurately predicted, it is assumed that the other results such as deformation, normal stress, von Mises stress, elastic strain, plastic strain, total strain, etc. are also valid.
5. The Menetrey-Willam model was able to predict the flexural behavior of reinforced UHPC beams by either using the full or quarter geometry model.
6. The Microplane model can predict the shear behavior of reinforced UHPC beams under a 3-point load experimental test.
7. Overall, when U-B3 and U-B5 are considered as control samples for flexure and shear respectively, U-B4 and U-B6 show increased load capacities, as indicated in Table 3. This increase is due to the higher bottom reinforcements provided in U-B4 and U-B6, making these beams significantly stiffer than the control samples U-B3 and U-B5.

Conflict of interests

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding

This research received no external funding.

Data availability statement

Data generated during the current study are available from the corresponding author upon reasonable request.

References

[1] Ahmad S, Sharif AM, Al-Osta MA, Al-Zahrani MM, Sharif AM, Al-Huri MA (2023) Flexural performance of pre-damaged RC beams strengthened with different configurations of UHPFRC layer–experimental and analytical investigation. Structures 48:1772-1787.
[2] ANSYS (2022) Material Reference. ANSYS Mechanical APDL Material Reference.
[3] Argyris JH (1969) ASKA—Automatic System for Kinematic Analysis: A universal system for structural analysis based on the matrix displacement (finite element) method. Nuclear Engineering and Design 10(4):441-455.
[4] Bazant ZP, Byung HO (1985) Microplane model for progressive fracture of concrete and rock. ASCE.
[5] Bazant ZP, Gambarova PG (1984) Crack shear in concrete: crack band microplane model. Journal of Structural Engineering 110(9):2015-2035.
[6] Bazoant ZP, Pijaudier-Cabot G (1989) Measurement of characteristic length of nonlocal continuum. Journal of Engineering Mechanics 115(4):755-767.
[7] Binnur GK (2022) FEM_2_Basic_Steps_in_FEM_Shape_Functions. Course note, Department of Mechanical Engineering, Dokuz Eylul University.
[8] Chen WF, Saleeb AF (2013) Constitutive Equations for Engineering Materials: Elasticity and Modeling. 1st Edition, Volume 37, Elsevier.
[9] Dahish HA, Bakri M (2022) Flexural behavior of RC composite beams with ultrahigh performance fiber reinforced concrete layer using finite element modeling. International Journal of GEOMATE 22(93):75–82. https://doi.org/10.21660/2022.93.3089
[10] El-Helou RG, Graybeal BA (2022) Flexural behavior and design of ultrahigh-performance concrete beams. Journal of Structural Engineering 148(4).
[11] Grøstad Ø, Sandberg E (2018) Analyzing UHPFRC Beams with the help of ANSYS. Master's thesis, Universiteteti Agder, University of Agder.
[12] Jiang H, Zhao J (2015) Calibration of the continuous surface cap model for concrete. Finite Elements in Analysis and Design 97:1-19.
[13] Jirásek M (1999) Numerical modeling of deformation and failure of materials. Unpublished lecture notes. Available at http://power2.fsv.cvut.cz/~milan/course.html
[14] Le Bellégo C, Dubé JF, Pijaudier-Cabot G, Gérard B (2003) Calibration of nonlocal damage model from size effect tests. European Journal of Mechanics-A/Solids 22(1):33-46.
[15] Leukart M, Ramm E (2003) A comparison of damage models formulated on different material scales. Computational Materials Science 28(3-4): 749-762.
[16] Menetrey P (1994) Numerical analysis of punching failure in reinforced concrete structures. Thesis, EPFL.
[17] Moaveni S (2015) Finite Element Analysis: Theory and Application with ANSYS. Pearson Education.
[18] A Hands-on Introduction to Engineering Simulations. Available at https://learning.edx.org/course/course-v1:CornellX+ENGR2000X+1T2018/home (Accessed 2021).
[19] https://s100.copyright.com/AppDispatchServlet?publisherName=ELS&contentID=S1877705817360629&orderBeanReset=true (Accessed 26 Aug 2024).
[20] Russell HG, Graybeal BA, Russell HG (2013) Ultra-High-Performance Concrete: A State-of-the-Art Report for the Bridge Community (No. FHWA-HRT-13-060). United States Federal Highway Administration. Office of Infrastructure Research and Development.
[21] Solhmirzaei R, Kodur VKR (2017) Modeling the response of ultra-high-performance fiber reinforced concrete beams. Procedia Engineering 210:211-219.

[22] Solhmirzaei R, Kodur V (2021) A numerical model for tracing structural response of ultra-high performance concrete beams. Modelling 2(4):448-466.
[23] Valente R, Alves G, Pacheco P Design of a pre-stressed bridge deck with ultra-high-performance concrete (UHPC).
[24] Willam K J (1975) Constitutive model for the triaxial behaviour of concrete. Proc. Intl. Assoc. Bridge Struct Engrs 19:1-30.
[25] Xenos D, Grégoire D, Morel S, Grassl P (2015) Calibration of nonlocal models for tensile fracture in quasi-brittle heterogeneous materials. Journal of the Mechanics and Physics of Solids 82:48-60.
[26] Yoo DY, Banthia N, Yoon YS (2017) Experimental and numerical study on flexural behavior of ultra-high-performance fiber-reinforced concrete beams with low reinforcement ratios. Canadian Journal of Civil Engineering 44(1):18-28.
[27] Zreid I, Kaliske M (2014) Regularization of microplane damage models using an implicit gradient enhancement. International Journal of Solids and Structures 51(19-20):3480-3489.
[28] Zreid I, Kaliske M (2016) An implicit gradient formulation for microplane Drucker-Prager plasticity. International Journal of Plasticity 83:252-272.
[29] Zreid, I, Kaliske M (2018). A gradient enhanced plasticity–damage microplane model for concrete. Computational Mechanics 62(5):1239-1257.