Cement and Concrete Research 106 (2018) 130–143

Contents lists available at ScienceDirect

Cement and Concrete Research

journal homepage: www.elsevier.com/locate/cemconres

A discrete-continuum coupled finite element modelling approach for fibre reinforced concrete

H. Zhang[a], Y.J. Huang[b], Z.J. Yang[a,*], S.L. Xu[a], X.W. Chen[c]

[a] *College of Civil Engineering and Architecture, Zhejiang University, 310058, China*
[b] *Institute of Architectural Design and Research of Zhejiang Province, 310006, China*
[c] *The State Key Lab of Explosion Science and Technology, Beijing Institute of Technology, 100081, China*

ARTICLE INFO

Keywords:
Fibre reinforced concrete
Cohesive crack model
Damaged plasticity model
Damage and fracture
Interfacial debonding
Finite element model

ABSTRACT

Fibre reinforced concrete (FRC) exhibits complicated failure modes such as fibre breakage, mortar cracking, crushing and spalling, fibre-mortar interfacial debonding, depending on many material properties, geometric dimensions, boundary and loading conditions. Most existing numerical models are unable to reproduce these failure modes that may occur simultaneously or sequentially in a specimen, mainly due to difficulties in generating finite element meshes with a large number of randomly-oriented fibres. Herein we develop a discrete-continuum coupled finite element modelling approach for FRC materials capable of effectively simulating all the major failure modes. The continuum damaged plasticity model is used to simulate damage and fracture behaviour of the mortar, while debonding of fibre-mortar interfaces is modelled by nonlinear cohesive interfacial elements. Unique techniques are devised to generate conforming meshes between fibres and the surrounding mortar so that the randomly-oriented fibres are easily modelled. The modelling approach is validated by simulating single fibre pullout tests with different inclination angles, notched and non-notched direct tensile tests and three-point bending beam tests with randomly-distributed multiple fibres.

1. Introduction

Traditional concrete structures are susceptible to fracture and sudden failure, mainly due to the low tensile strength and high brittleness of the concrete material consisting of aggregates, cement paste (mortar) and pores. To tackle this problem, fibres made from steel, polymer, carbon and glass etc. [1–5], are widely used to reinforce concrete (i.e., fibre reinforced concrete (FRC)), making use of high tensile strength and bridging capability of fibres to resist initiation and propagation of cracks. Nowadays FRC with different mixtures is routinely used in engineering structures with much improved load-carrying capacity, ductility and durability. However, the higher cost of FRC materials than traditional concrete still prevents their potential from being fully realised, especially for high-end FRC materials, such as the ultra-high performance fibre reinforced concrete (UHPFRC) with improved strength capacity (compressive strength over 150 MPa and tensile strength over 8 MPa), fracture toughness and ductility [6–10]. Optimization of the materials at micro/meso scales is thus highly desired so that the key parameters such as the fibre volume fraction can be fine-tuned to meet users' requirements in engineering practice. This is only possible after a thorough understanding of the materials' micro/meso-scale mechanical behaviour is acquired by fundamental research.

Both extensive experiments and numerical simulations have been carried out for the above purpose. Herein we focus on the latter at the length scale of micrometre to millimetre (meso-scale). At such scales, an FRC material is a composite comprising mortar matrix, reinforcing fibres, fibre-mortar interfaces and pores. Its mechanical properties and failure behaviour are thus very complicated, depending on many factors such as the strength, stiffness and volume/weight fractions of the constituents, the length, shape and distribution of fibres, the size, number and distribution of pores, and inter-phasic interactions such as bond-slip between fibres and mortar. Depending on the relative values of these factors, various failure modes with distinguished characteristics can take place, such as fibre breakage, mortar cracking, crushing and spalling, and fibre-mortar interfacial debonding. However, most reported numerical models have assumed FRC as a homogeneous material whose constitutive laws are obtained from laboratory tests of macro-scale samples. These models are mostly based on continuum mechanics, such as the damaged plasticity model [9], the microplane model [11], the failure surface model [12] and the stress transfer-based model [13]. Because the meso/micro-scale features such as fibres and interfaces are not considered explicitly, these homogeneous models cannot predict

---

* Corresponding author.
E-mail address: zhjyang@zju.edu.cn (Z.J. Yang).

https://doi.org/10.1016/j.cemconres.2018.01.010
Received 28 April 2017; Accepted 11 January 2018

the fundamental failure mechanisms.

At the meso/micro-scale, many nonlinear finite element models have been developed to study the single fibre pullout tests (SFPTs), because there is only one layer of matrix-fibre interface that can be relatively easy to simulate and experimental data are widely available for comparison. For example, for SFPTs with the fibre aligned with the loading direction, Ellis et al. [14] developed finite element (FE) models with the matrix simulated by the Drucker-Prager plastic model and the interfacial transitional zone (ITZ) by the Coulomb friction law. The matrix-fibre interaction was modelled by contact surfaces between the ITZ and the fibre. As the ITZ was very thin (50 μm), very fine local meshes were used. Jia et al. [15] simulated SFPTs with matrix reinforced by carbon nanotubes using cohesive elements to model the fibre-matrix interface. The matrix damage was not modelled. For SFPTs with the fibre inclined to the loading direction, fibre bending, local friction and matrix spalling can all occur. Leung and Li [16] modelled the fibre by beam element and the matrix as linear springs. The matrix spalling was governed by a failure strain criterion and the fibre-matrix interaction was considered smooth or frictional by an iterative contact algorithm. Zhang and Yu [17] proposed a model to capture the entire pullout behaviour, using nodal contact elements to describe the bonding force and friction independently. The matrix was assumed elastic with a spalling criterion proposed by Laranjeira [18] to determine the size of the spalled wedge.

Although the micro/meso-FE models for SFPTs are indispensable to investigate the fundamental failure mechanisms (especially the fibre-matrix debonding and friction), they are insufficient to simulate real FRC specimens with a large number of randomly-oriented fibres and thus very complicated nonlinear fibre-matrix interfacial behaviour which also interacts with matrix cracking and crushing. Indeed, many of the FE models with multiple fibres have so far been developed for numerical homogenisation of elastic properties, either based on computer-generated fibres with random orientations [19] or more recently, converted directly from realistic images, such as from X-ray computed tomography scanning [20]. Nonlinear FE models with multiple fibres are still rare. For example, Cunha et al. [21] used the smeared crack model and Yu et al. [23] assumed elastic behaviour for the matrix and both modelled beam-bending tests with a single cohesive crack in the middle. The fibre-matrix interaction was indirectly modelled by equivalent tensile stress-strain constitutive laws of fibres transformed from load-slip curves of SFPTs. However, the equivalent transformation remains empirical and arguable. In particular, for the fibre embedded length, which is a key parameter, Cunha et al. [21] used a quarter of fibre length based on the analytical study of Stroeven and Hu [22], and Yu et al. [23] used the total fibre length assuming uniform stress distribution along the fibre. It is also difficult to interpret the resultant stress in a fibre, as it mixes the effects of elongation of the fibre and bond-slip on the fibre-matrix interface.

Apart from the conventional FE models, Bolander et al. [24] developed a lattice model in which the fibres were modelled by spring elements crossing the boundaries of matrix cells. The spring stiffness was reduced to indirectly account for interfacial debonding. Kang et al. [25] further developed an improved lattice model by uniformly distributing the pull-out force from a SFPT along the embedded length of all the fibres crossing a crack, which was unrealistic. FE models based on the partition of unity [26] and on the immersed boundary method [27] are also developed for FRC with multi-fibres, so that the fibre mesh does not need to conform to the matrix mesh. Although these models are computationally efficient or easy to generate and implement, they cannot directly simulate the micro/meso-scale failure mechanisms, such as the discrete interfacial debonding, fibre bending, yielding and snubbing effect of inclined fibres.

This study is aimed at developing an innovative, easy-to-implement, discrete-continuum coupled finite element modelling approach, which is capable of simulating all the possible failure mechanisms in FRC with a large number of fibres at the micro/meso-scales. In this approach, the material phases, i.e., the fibres, the matrix and the fibre-matrix interfaces are individually and explicitly represented. The fibres are simulated by beam elements with elastic-plastic properties, the mortar matrix by solid elements with continuum damaged plasticity mechanics, and the fibre-matrix interfaces by zero-thickness cohesive elements with softening bond-slip relations. The cohesive elements are arranged in a unique way so that the nonlinear interfacial debonding and friction can be modelled discretely. This also makes mesh generation very simple, even for models with a large number of randomly-distributed fibres. The damaged plasticity model can also capture cracking, crushing and spalling in the mortar matrix. Therefore, all the major failure mechanisms at micro/meso-scales can be simulated by the proposed approach, depending on the relative material properties of different phases. This capability is particularly important for an FRC model, because all the failure mechanisms may occur simultaneously or sequentially at different locations due to random distribution of fibres. Moreover, the ambiguous assumptions and reliance on the load-slip curves from macro-scale SFPTs, required by many existing models for interfacial bond-slip simulation, are now not needed. As realistic constitutive laws are assigned for each phase, all the results have clear physical interpretation, and parametric studies can be carried out to optimise the key parameters at the material level.

The paper is organised as follows. The methodology is presented in Section 2, including the mesh generation procedure implemented by a Python script in ABAQUS and a MATLAB code. The constitutive laws for the matrix, fibres and the interfaces are also discussed. In Section 3, a SFPT is simulated as a benchmark and the results are critically compared with experimental data, followed by an extensive parameter study. In Section 4, the developed model is further validated against beam bending and direct tensile tests with random fibres. The main conclusions are drawn in Section 5. It should be noted that the FRC materials investigated in this paper have no coarse aggregates, but they can be added and modelled with ease [28,29].

2. Methodology

2.1. Mesh generation

Fibres are randomly distributed in the domain. Random numbers are generated and used to define the first fibre's centre point and its orientation. The two end points of the fibre are then calculated as the fibre length is known. The fibre will be shortened if it intersects with the domain boundary. The next fibre is then generated in the same way until the given volume fraction of fibres is reached.

A two-step scheme is devised to generate the mesh. In the first step (see Fig. 1a), all the fibres are set as Part A and the matrix as Part B in ABAQUS. A Boolean operation is then performed to merge the fibres into the matrix. This operation makes the fibre geometries as boundaries of the matrix so that the fibres share the same nodes as the matrix after meshing (i.e., conforming). The matrix mesh in Fig. 1b consists of 4-noded plane-stress isoperimetric elements (CPS4R in ABAQUS) and the fibre mesh consists of two-noded Timoshenko beam elements (B21 in ABAQUS). The beams elements are used herein because steel fibres are comparatively stiff, and bending deformation cannot be neglected [30]. This step is implemented by a Python script in ABAQUS. In the second step, zero-thickness cohesive elements (COH2D4 in ABAQUS) are inserted between pairs of fibre and matrix elements to model the fibre-matrix interface, using a simple MATLAB code (Fig. 1b). The cohesive elements are arranged in a unique way that both the fibres and interfaces are deformed in the plane but the fibres appear “floating” on the matrix. This avoids the use of very fine local meshes in the matrix surrounding the thin fibres if the fibre nodes are also located in the plane. Fig. 1c shows a special case when two fibres intersect with each other. At the intersection, three nodes with the same coordinates are used, i.e., N1 of fibre 1, N2 of fibre 2 and N3 of the matrix.