Engineering Structures 51 (2013) 73–87

Contents lists available at SciVerse ScienceDirect

Engineering Structures

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

FE modelling of RC frames with masonry infill panels under in-plane and out-of-plane loading

Alireza Mohyeddin [a,*], Helen M. Goldsworthy [b], Emad F. Gad [c]

[a] School of Engineering, Edith Cowan University, Joondalup, WA 6027, Australia
[b] Dept. of Infrastructure Engineering, The University of Melbourne, Parkville, VIC 3010, Australia
[c] Faculty of Engineering and Industrial Sciences, Swinburne University of Technology, Hawthorn, VIC 3122, Australia

ARTICLE INFO

Article history:
Received 28 March 2012
Revised 16 January 2013
Accepted 20 January 2013
Available online 26 February 2013

Keywords:
Finite element
ANSYS
Infill-frame
Strut modelling
Masonry
Reinforced-concrete
In-plane
Out-of-plane

ABSTRACT

This paper gives a detailed presentation of a generic three-dimensional discrete-finite-element model that has been constructed for reinforced-concrete frames with masonry infill using ANSYS. Appropriate experimental data available from the literature are utilised to verify the model. The reasons behind some of previously observed damage to infill-frames are given. A simple method is proposed to overcome convergence issues which are related to the Newton–Raphson algorithm. It is shown that the model can be employed to predict the behaviour of the infill-frame over a wide range of drift, and to interpret its response at various stages of in-plane or out-of-plane loading.

1. Introduction

The structural interaction between a structural frame and its masonry infill panel was first appreciated in the late 1930s and early 1940s. Polyokov [1] referred to studies in 1938–1939 on the Empire Building which showed that the actual in-plane stiffness of the infill-frame was more than the (calculated) stiffness of the frame. An interesting reference was made to the studies by Li Onishchik in 1937 and 1939; Onishchik concluded that it would be possible to rely on the capacity of an infill panel in the design of a frame by considering the panel as a "compressed diagonal strut". Holmes [2] proposed an equivalent cross sectional area of such a strut for the first time. The concept of diagonal struts has been widely investigated; a variety of models have been proposed and applied for the purpose of structural analysis (e.g. [3–26]). For a comprehensive comparison between different strut models one can refer to Crisafulli et al. [27], Asteris et al. [28], and Chrysostomou and Asteris [29].

Despite the similarity between the crack patterns of the infill-frame when subject to in-plane and out-of-plane loading, the interaction between the two has largely been ignored. Angel [9], Flanagan and Bennett [30] and Calvi et al. [31] are within rare examples of the studies considering the interaction between the two loadings.

The Finite Element (including Discrete Finite Element) method has been used by many researchers for the analysis of masonry and infill-frame structures. Karamanski [32], Mallick and Severn [33], Riddington and Stafford Smith [34], Page [35], Arya and Hegemier [36], Liauw and Kwan [37], Dhanasekar et al. [38], Ali and Page [39], Riddington and Gambo [40], Rots [41], Lourenco and Rots [42,43], Lotfi and Shing [44,45], Mehrabi [10], Crisafulli [14], Asteris [46,47] are examples of such studies.

In the recent studies by Stavridis and Shing [48,49] and Koutromanos et al. [50], the previous constitutive material model and interface elements proposed by Lotfi and Shing [44,45] were further developed with the addition of interface elements within the frame members to account for shear failure of RC members. Similar to Mehrabi [10] this was implemented within the finite-element code FEAP. Seah [51] developed a FE model creating a masonry panel by using the failure surfaces for masonry proposed by Lourenco [52]. Software "ALGOR" was used by Al-Chaar [16] and D'Ayala et al. [53] for their FE modelling. Mohebkhah et al. [54] proposed a two-dimensional model developed using the discrete element software UDEC for infill-frame analysis. Moghadam and Goudarzi [55] used the code "ABAQUS" and applied the dynamic explicit method.

* Corresponding author. Tel.: +61 8 6304 5201; fax: +61 3 6304 5811.
E-mail addresses: a.mohyeddin@ecu.edu.au (A. Mohyeddin), helenmg@unimelb.edu.au (H.M. Goldsworthy), egad@swin.edu.au (E.F. Gad).

0141-0296/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.engstruct.2013.01.012

ANSYS has been used by some researchers for FE modelling of concrete beams [56–59]. Aliaari [60] used ANSYS in a simple manner to analyse infill-steel frames under in-plane loading. Most of the researchers who attempted using ANSYS for concrete modelling have assumed that concrete behaves linearly up to the point where it crushes and/or cracks. In fact, the material can exhibit considerable nonlinearity in its behaviour prior to crushing.

This paper uses ANSYS in order to develop a three-dimensional FE model of infill-RC frames at a microlevel. First, the model is verified for reinforced-concrete and masonry modelling separately. The two strategies are next combined in the form of an infill-frame and verified against experimental results available from the literature. The advantages as well as limitations of the present model are discussed. Based on the results of the constructed FE models, the reasons behind some of previously observed damage to infill panels are discussed. It is shown that the model can be employed to interpret the response of the infill-frame under in-plane or out-of-plane loading over a wide range of drift. This will be a useful tool for parametric/sensitivity analysis of the infill-frame such that the same model can be used for the assessment of the infill-frame under both in-plane and out-of-plane loading.

2. Reinforced-concrete modelling

2.1. Failure surface and concrete material model for compression

Reinforced-concrete and masonry materials are modelled using the SOLID65 finite element from ANSYS. This element has smeared reinforcement and smeared cracking capabilities [61] and uses the failure surface proposed by Willam and Warnke [62]. The shortcoming of the SOLID65 element is that it assumes a linear stress–strain relationship. For this reason, it is necessary that this concrete model be combined with another nonlinear model within ANSYS in order to produce the nonlinear stress–strain relationship. Based on the discussions presented by Mohyeddin [63], the failure surface produced as the result of this approach is similar to that taken by Lotfi and Shing [44], except that in the present model the failure surface is further implemented in a three-dimensional stress space. Fig. 1 shows a select number of failure surfaces in a two-dimensional stress space. For comparison purposes, the failure surface proposed by CEB-FIP Model Code 90 [64] is also provided in this figure.

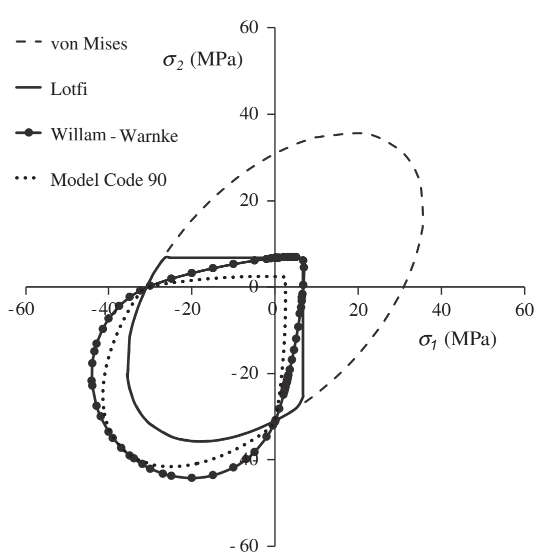

Fig. 1. A comparison between different failure surfaces for plane-stress state in concrete.

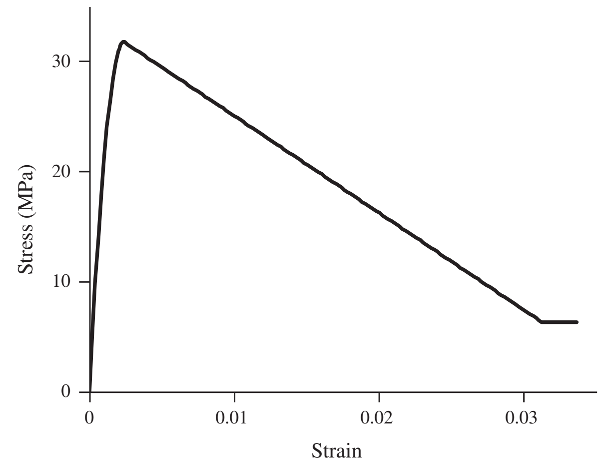

Fig. 2. Stress–strain relationship for concrete.

The uniaxial stress–strain relationship for confined concrete, known as the modified Kent and Park model, has been incorporated in the FE model constructed here. This model shows a good agreement with the experimental results [65,66] and offers a good balance between simplicity and accuracy [67]. Fig. 2 includes a graphical example of this equation for $f'_c = 30.9$ MPa, where f'_c is the specified/characteristic compressive strength of the concrete. This value is the concrete strength recorded in the experiment used for verification of reinforced-concrete FE modelling (Section 6).

The shear coefficient for open cracks, which controls the amount of shear transferred across an open crack, has been calculated based on the transverse reinforcement at each section [63].

2.2. Concrete material model for tension

The finite element used in this model has the smeared cracking capability. Cracking occurs at the integration points along three orthogonal directions based on the predefined tensile strength of the material. Although the concept of fracture energy is widely used in analytical models for concrete cracking [64,68–71], in ANSYS tensile cracks are not directly related to the fracture energy; instead cracking is defined by a single material parameter i.e. the tensile strength of concrete, f'_t. This being said, by knowing the mode I fracture energy, G_F^I (which is the area under the curve), and f'_t one can define the tensile stress relaxation (“T_c“ in Fig. 3) such that the energy dissipated under the stress–strain curve approximates the experimental value of G_F^I. The assumption of $6\varepsilon_{cr}$ for the descending branch of the stress–strain curve, where ε_{cr} is the strain corresponding to f'_t, is compatible with what Ali and Page [39] found for masonry.

2.3. Reinforcing rebars

A bi-linear stress–strain relationship is assumed for the steel material. The modulus of elasticity of steel, E_s, is assumed to be 200 GPa, and the secondary stiffness, E_2, (also known as the “tangent stiffness”) is approximately measured to be 2.5% of E_s [63]. As it is applicable to most metals, the von Mises failure surface with a total stress range of twice the yield stress (Bauschinger effect) is used here for the reinforcing steel.

3. Masonry modelling strategy

In the modelling strategy adopted in this research the mortar joint thickness is halved; each half is attached to the adjacent