11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

A NEW FOURIER–RELATED DOUBLE SCALE FINITE ELEMENT FOR MEMBRANE INSTABILITY PHENOMENA

Qun Huang[1], Heng Hu[1], Kun Yu[1], Michel Potier-Ferry[2], Salim Belouettar[3] and Gaetano Giunta[3]

[1] School of Civil Engineering, Wuhan University, 8 South Road of East Lake, Wuchang, 430072 Wuhan, PR China, Email: huheng@whu.edu.cn
[2] Laboratoire d'Etude des Microstructures et de Mécanique des Matériaux, LEM3, UMR CNRS 7239, Université de Lorraine, Ile du Saulcy, 57045 Metz Cedex 01, France
[3] Centre de Recherche Public Henri Tudor, 29, avenue John F. Kennedy, L-1855 Luxembourg-Kirchberg, Luxembourg

Key words: *Fourier series, Membrane, Instability, Wrinkling, Asymptotic Numerical Method.*

The paper presents a new Fourier-related double scale finite element to study membrane instability phenomena. Based on Von Karman plate equations, two kinds of FE numerical macroscopic models are developed by using the technique of slowly variable Fourier coefficients: 1) new reduced model, 2) pure membrane model by dropping the derivatives of envelop of wrinkles in the new reduced model. The results show that the first model is able to accurately and quickly predict critical load and describe post-buckling process, while the second one limits on qualitatively analyzing wrinkling phenomenon. We discover membrane instability strongly depends on boundary conditions. A dimensionless parameter K is defined that remains almost a constant near bifurcation point in rectangular membrane buckling which plays an important role in membrane buckling prediction and understanding. The established nonlinear systems are solved by the Asymptotic Numerical Method (ANM), which has the advantages of fast convergence and high computational efficiency compared with Newton-Raphson method. Finally, the new reduced model is implemented into the commercial finite element software ABAQUS as a user element (UEL) to simplify simulation process and use in more complex membrane structures.