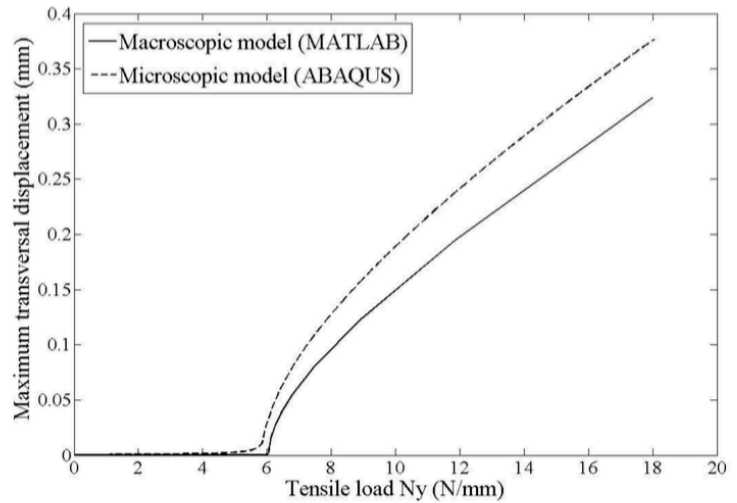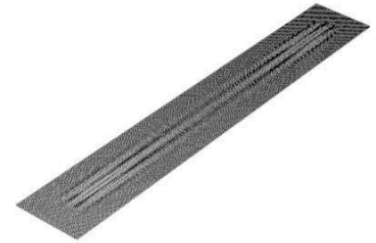**Fig. 2:** Bifurcation curves of the two models

Macroscopic model (MATLAB)   Microscopic model (ABAQUS)

Dr. Kodji Attipou

From: Kodjo Attipou, Heng Hu, Foudil Mohri, Michel Potier-Ferry and Salim Belouettar, "Analysis of buckling and wrinkling of plate under thermo-mechanical loading using a Fourier-related multi-scale approach", The 6th International Conference on Coupled Instabilities in Metal Structures, December 2012,

See:
http://emploi.techniques-ingenieur.fr/profil/kodjo-attipou

Experience:
1. Doctoral thesis on the topic of instability in composite plates
2. Simulation and modeling in Abaqus and MATLAB.
3. Using the Fourier method multiscale, digital asymptotic method for programming in finite element code.
4. Using the method for the complete Harlequin reduced-model coupling.

Education:
MS – 2009-2010 Paul Verlaine University Metz (materials science)
Ph.D – 2011-2014 Paul Verlaine University Metz (mechanics of materials)

Selected Publications:

Kodjo Attipou (1), Heng Hu (2), Foudil Mohri (3), Michel Potier-Ferry (3) and Salim Belouettar (1)
(1) Centre de Recherche Public Henri Tudor, 29, Avenue John F. Kennedy, L-1855 Luxembourg-Kirchberg, Luxembourg
(2) School of Civil Engineering, Wuhan University, 8 South Road of East Lake, 430072 Wuchang, PR China.
(3) LEM3, Laboratoire d'Etude des Microstructures et de Mécanique des Matériaux, UMR CNRS 7239, Université de Lorraine, Ile du Saulcy, 57045 Metz Cedex 01, France.
ABSTRACT: In this paper, we study the temperature's effect on a plate under global tensile stress. For this aim, the Fourier double scale method is used in order to develop a macroscopic model that is a generalized continuum. This model couples local and global instabilities in wrinkling phenomena. The advantage of this technique is to remain valid away from the bifurcation point while former techniques such as Landau-Ginzburg theory are valid only close to the bifurcation point. Due to its efficiency, this method is extended to plate structures. A finite element formulation of the plate model is made. The model developed was implemented using MATLAB code and a validation test has been conducted, comparing to ABAQUS simulation.