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

CHANG-HSIEH-CHEN LOW-REYNOLDS *k-ε* TURBULENCE MODEL– ADAPTATION TO STUDY THE FLOW OF CONCENTRATED PULP SUSPENSIONS IN PIPE

C. Cotas[1], F. Garcia[1], P. Ferreira[1], P. Faia[2], D. Asendrych[3] and M.G. Rasteiro[1*]

[1]Centre for Chemical Engineering and Forest Products, Chemical Engineering Department, University of Coimbra, Portugal
[2]Electrical and Computers Engineering Department, University of Coimbra, Portugal
[3]Faculty of Mechanical Engineering and Computer Science, Częstochowa University of Technology, Poland
[*] e-mail: mgr@eq.uc.pt

Key Words: *non-Newtonian fluid, Chang-Hsieh-Chen turbulence model, damping function, drag reduction.*

Computational Fluid Dynamics (CFD) tools can be used to improve the design of industrial equipment avoiding the use of empirical correlations. In the pulp and paper industry it is very common to base the design of industrial equipment on empirical correlations that have limitations which, usually, lead to oversized and low efficiency equipment.

The present study investigated the turbulent flow of concentrated *Eucalyptus* pulp suspensions in pipes using the commercial software Ansys Fluent for its simulation. A pseudo-homogenous approach was followed in this work, considering the pulp suspension viscosity a function of shear rate, non-Newtonian behaviour, as obtained experimentally. Furthermore, a water annulus at the pipe wall surrounding the flow core was taken into account in the model. The CHC (Chang-Hsieh-Chen) low-Re *k-ε* turbulence model [1] was selected to study the fiber suspension flow when a *drag reduction* effect is present. In the literature, only a few studies are dedicated to concentrated pulp suspensions flows. The application of low-Re *k-ε* turbulence models has been presented to study polymers flow but not pulp fibers suspensions flow.

Different tests were carried out to investigate the applicability of the CHC turbulence model with a damping function adapted from literature [2], where the damping function was applied to study the flow of a power-law fluid in a pipe. A series of simulation tests was carried out to investigate the influence of some constant parameters in the damping function, on the numerical results.
The model parameters were tuned based on experimental information available in the literature and then validated by comparing the numerical results for pressure drop with those obtained experimentally. The modification introduced on the original CHC turbulence model allowed a good correspondence between the velocity profiles reported in literature and those obtained numerically. In fact, a different behaviour from that obtained for a Newtonian fluid, for the dimensionless velocity as a function of dimensionless distance to the pipe wall, was reported in literature for fiber suspensions flow, showing a peculiar S-shaped profile near the

wall.

In the present study, when the velocity profile followed the tendency reported above, a better agreement between the numerical pressure drop and the experimental one was obtained. Moreover, the *drag reduction* effect was correctly reproduced by modifying the diffusivity term and source-term (non-Newtonian behaviour and damping function).

REFERENCES

[1] K.C. Chang, W.D. Hsieh and C.S. Chen, A modified low-Reynolds-number turbulencemodel applicable to recirculating flow in pipe expansion, Vol. **117**, pp. 417-423, 1995.

[2] M.R. Malin, Turbulent pipe flow of power-law fluids, Vol. **24**, pp. 977-988, 1997.