Abstract:

Aqueous foam flow presents original mechanical properties that can give important place to a number of industrial applications: heat exchangers in power plants (nuclear engineering, petroleum engineering, coal power plants, etc.), food industry, cosmetic industry, fire extinguish, etc. The improving of these applications can only be achieved by the understanding and simplification of the foam flow internal structure. In foam flow, an important parameter is the behavior of this fluid. In fact, the pressure drop inside ducts, directly linked to the wall shear stress, plays an important role.

In this study, we characterize firstly experimentally the behavior of this aqueous foam flow along a square channel in presence of a circular cylinder as an obstacle. These results will be compared against numerical simulations done with Computational Fluid Dynamics (Ansys CFX). The cylinder is located in the middle of the channel with a square section of 21x21 mm². Experimental results are obtained in order to well understand the singularity influence over the foam flow and its texture behind the obstacle. The objective, is to validate the use of CFD in this kind of non-Newtonian fluid in order to predict the pressure losses, the wall shear stress and the flow dynamics. The Particle Image Velocimetry (PIV) Technique will be used to obtain the dynamics of the wet foam flow downstream the obstacle. A series of pressure outlets allows to obtain the singularity pressure losses along the channel length. Foam flow considered as a highly complex fluid. However, from a macroscopic point of view, the interaction between liquid films and air bubbles can be simplified by the use of non-Newtonian pseudo-fluid models. These ones can represent the internal deformations of fluid elements when high shear is applied. It is taken into consideration an established flow that behaves as a piston or a block, with a foam velocity around 2 cm/s and a void fraction of 70%. As for the numerical simulation, a Bingham fluid is used in order to extrapolate the flow behavior predictions and compared them against the experimental results. These comparisons show that the CFD approach is able to recreate the behavior of a wet foam flow over a circular cylinder.

This study allows to better understand physical phenomena in this kind of complex fluid flow. Pressure losses and wall shear stress, which can be clearly showed in this work, permit to well understand and avoid fouling problems inside ducts in coal power plant circuits, for example.

Keywords: Foam flow, complex fluid, PIV, Polarography, CFD, pressure drop, wall shear stress, fouling.

Acknowledgement: The scientific work was supported by the laboratory TEMPO (Team DF2T) of the University of Valenciennes (France). This support is gratefully acknowledged.

* Corresponding author:
E-mail: Fethi.Aloui@univ-valenciennes.fr Tel: +33 3 27 51 19 62, Fax: +33 3 27 51 19 61