



Gas-Liquid Foam Through Straight Ducts and Singularities: CFD Simulations and Experiments

Rogelio Chovet, Fethi Aloui, Laurent Keirsbulck

► To cite this version:

Rogelio Chovet, Fethi Aloui, Laurent Keirsbulck. Gas-Liquid Foam Through Straight Ducts and Singularities: CFD Simulations and Experiments. ASME 2014 12th International Conference on Nanochannels, Microchannels, and Minichannels, Aug 2014, Chicago, United States. 10.1115/FEDSM2014-21190 . hal-03474238

HAL Id: hal-03474238

<https://uphf.hal.science/hal-03474238>

Submitted on 7 Jul 2022

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



Distributed under a Creative Commons Attribution 4.0 International License

GAS-LIQUID FOAM THROUGH STRAIGHT DUCTS AND SINGULARITIES: CFD SIMULATIONS AND EXPERIMENTS

Rogelio Chovet

rogelio.chovet@univ-valenciennes.fr

Fethi Aloui *

fethi.aloui@univ-valenciennes.fr

Laurent Keirsbulck

laurent.keirsbulck@univ-valenciennes.fr

University of Lille North of France - University of Valenciennes (UVHC)
TEMPO Laboratory, DF2T, Campus Le Mont Houy
59313 Valenciennes Cedex 9 - France

ABSTRACT

Some industrial processes are associated with the flow of aqueous foams inside horizontal channels. Examples are found in the oil, food and cosmetic industries. This type of flow presents an important pressure loss, originated from the shear stress exerted by the channel walls. Foam flow is one of the most complex fluids. In a macroscopic point of view, the physical-chemical interaction between the bubbles can be related to some non-Newtonian models (Bingham law, power law, etc.) or an apparent viscosity. These last can represent the internal deformations of fluid elements when shear stress is applied. An experimental facility able to create this type of flow is not so easy to design. Many parameters must be taken into consideration. So, Computational Fluid Dynamics (CFD) constitutes an ideal technique for analyzing this kind of problem. The aim of this study is to validate the use of Computational Fluid Dynamics in order to correctly predict the pressure losses and the velocity fields of a foam flowing through a straight channel and singularities (fence and half-sudden expansion). Simulations for a realistic scenario: two-phase flow, change in the surface tension, bubble size, were undertaken. Obtained results showed that simulations are not able to accurately reproduce for such a complex fluid, the important aspects of this study, such as the pressure losses and the velocity fields. Therefore, an approximation to a Bingham fluid was made. For a foam flow quality of 70% and a velocity of 2 cm/s, the numerical results are justified by experimental evidence. Experiments have been done and predictions for the flow

behavior are extrapolated. Results show that the software is able to recreate the behavior of foam flow through a straight channel and singularities. However, this approach is extremely sensitive to the choice of several parameters, like the apparent viscosity, the yield stress, the viscosity consistence, etc.

NOMENCLATURE

Roman letters

\vec{e}_x	Unit vector of the Ox axis
P	Static pressure (Pa)
P_0	Static pressure at the exit of the duct (Pa)
u	Velocity component in the x direction (cm/s)
\vec{u}	Velocity vector (cm/s)
v	Velocity component in the y direction (cm/s)
x	Horizontal tranverse coordinate (mm)
y	Vertical transverse coordinate (mm)

Greek letters

γ	Yield ($1/s$)
k	Viscosity Consistency (Pa.s)
ρ	Density (kg/m^3)
τ	Stress (Pa)
τ_c	Yield Stress (Pa)

INTRODUCTION

The word foam originates from the medieval german *veim*. It is a substance that is formed by trapping pockets of gas in a

*Address all correspondence to this author.

liquid or a solid. In most of them, the gas volume is larger than the liquid one. The liquid is located in thin films separating the regions of gas. They are in general opaque, remarkably stable, and even elastic, as is evident for the shaving foam. The mechanical properties of foams (low density, high specific surface, disjoining pressure, foam-wall viscous friction) are important in several industrial applications. In oil recovery, for example, foam is used efficiently to drag petroleum out of porous media while augmenting the well pressure. The improve of these industrial processes will only be achieve with a better understanding of the foam rheology and accurate aproximations of foam flow behavior.

Rheology is the study of the deformation and flow of *complex* fluids that can exhibit both liquid and solid behavior. Although foams are mostly compose of gas, its rheological properties can be assimilated to those of a complex fluid [1]. Aqueous foams typically exhibit a combination of the following three types of behavior:

Elastic: Foam stores up mechanical energy, later on it releases it to return to its initial state.

Plastic: Beyond a certain stress point the deformation is irreversible and the foam will acquire and retain a new shape.

Viscous: The foam flows like a viscous liquid, dissipating an amount of energy that depends on the deformation rate.

This rheological behavior is a consequence of the high interfacial area [2]. Due to the many different length- and time-scales, foam rheology is particularly difficult to model. For the macroscopic scale (the foam sample is considered as a continuous medium), constitutive laws describe the relation between stress, deformation and flow. If we gradually increase the strain applied to the foam, the elastic stress becomes a non-linear function of the strain and in the end the material yields. in order to model the *steady* flow of materials that behave both as a solid or a liquid (depending on the applied stress) over a large range of strain rates, we must account for viscous stresses. The Bingham model is often used, for this kind of fluids the rheological behavior is expressed in Eqn. (1).

$$\begin{aligned} \gamma &= 0 & \text{if } \tau < \tau_y \\ \tau &= \tau_c + k\gamma & \text{if } \tau > \tau_c \end{aligned} \quad (1)$$

In order to study the foam flow behavior several approaches have been used. Theoretical studies have been limited because of the complexity of all the parameters involved in foams physycs and mathematics (plateau border stability, disjoining pressure, coalescence, drainage, ripening, etc.). Despite the complexity, important results are available [3]. The experimental approach has also numerous disadvantages, from the measurement methods to the foam flow stability regulation [4–6].

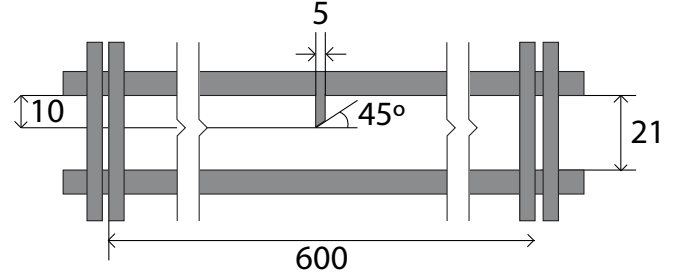


FIGURE 1: LATERAL VIEW OF THE FENCE [mm]

For all these reasons numerical approach seems more appropriate. There have been some foam simulations. However, these ones were focused in different subjects: foam generation or deterioration, physical properties, complex rheological behavior and for heat exchanging intentions [7–10]. Also, none of them have made an intensive study over an horizontal foam flow and compared both, experimental and numerical, results.

For this analysis, Computational Fluid Dynamics, CFD, is an appropriate method for obtaining and approximation of the foam behavior when flowing through a straight channel and singularities (fence and half-sudden expansion). CFD allows the consideration of different parameters, for non-Newtonian flows, such as velocity, pressure losses, shear stress, parameters that directly affect the flow regime, fluid mobility, and pressure profile along the channel. In particular, conclusion will be based in the evolution of the velocity and pressure profiles along the channel and in presence of singularities, then they will be compared against the experimental results obtained for the same conditions.

GEOMETRY AND NUMERICAL PARAMETERS

The dimensions of the numerical geometry were related to the experimental ones [4, 5]. The length of the channel variates as a function of the singularity used. For the square configuration it is 3.13 m, for the half-sudden is 3.87 m and for the fence 3.41 m. The inlet of the channel has a square section of 21 x 21 mm². The *measurement section* is located 1.3 m downstream of the conducts input. At this level the two singularities were placed. The fence (Figure 1) presents a height of 10 mm and an escape angle of 45°. As for the half-sudden expansion (Figure 2), its upstream has the same square section as the fence (21 x 21 mm²). However, with a section ratio of 0.5 the downstream area is 21 x 42 mm².

The coordinate systems for both, the fence and the half-sudden expansion were located at the height of the singularity. Therefore all measurement done downstream of it will have a positive value in the *x direction* and all done upstream a negative one.

In order to decrease the CPU time calculations and require-

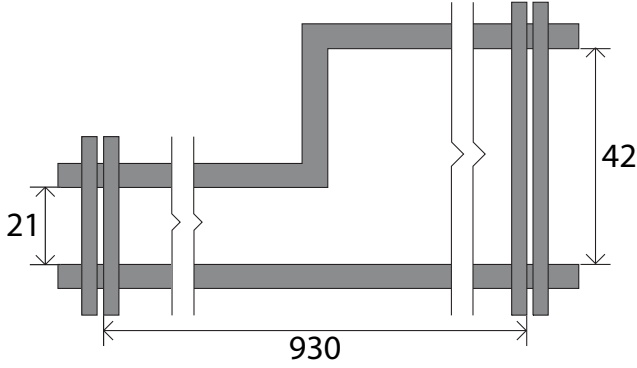


FIGURE 2: LATERAL VIEW OF THE HALF-SUDDEN EXPANSION [mm]

ments, all three cases were assumed to have a 2D geometry. Therefore, the depth of the geometry was assumed as infinite (x coordinate).

For the grid construction, mesh seeds are located in the axial and vertical direction of the models, then hexahedral elements are generated. Hexahedral elements allow better results than tetrahedral ones in terms of computing time and accuracy; therefore, the former types of elements are selected for this study.

For each geometry a small refinement was done for the interest regions, at the fence and at the expansion. A grid was considered adequate when the average velocity keeps a relative difference lesser than 0.1% when compared with the results obtained by using a refined grid with the double number of elements. For this analysis, refinement is done by altering the size of the element through the whole geometry. The mentioned average velocity is evaluated at the inlet of duct and at 1.5 m from it.

A foam inside a straight duct can present three different regimes, depending on its velocity [11]:

One-dimensional flow: For this regime the flow behaves as a whole, it moves as a block or a piston. The velocity vectors have only one uniform axial component in the flow direction.

$$\vec{u} = u \cdot \vec{e}_x = \text{cte} \cdot \vec{e}_x \quad (2)$$

Two-dimensional flow: It is obtained when the established flow has an axial velocity component that only depends of the y coordinate

$$\vec{u} = u(y) \cdot \vec{e}_x \quad (3)$$

Three-dimensional flow: it is obtained when the established flow velocity vector has an axial component that depends on

TABLE 1: RHEOLOGICAL PROPERTIES OF FLUIDS IN THIS STUDY

Geometry	ρ (kg/m ³)	τ_c (Pa)	k (Pa.s)
Half-Sudden ExpansionI	300.22	6.22	0.001
Fence	300.22	4.6	0.001

the z and y coordinates.

$$\vec{u} = u(y, z) \cdot \vec{e}_x \quad (4)$$

To establish the practical relevance of these considerations, the numerical simulations were done undertaking a regime close to the one (one-dimensional) used for the experimental measurements [4, 5].

Simulations with the Eulerian-Eulerian multiphase models were made. They took into consideration all the important parameters that a foam flow could have: a continuous phase and a disperse phase, diminution of the surface tension for the liquid phase, bubble size, drainage, wall lubrication force, drag force, etc. Despite all the data available inside the experimental results, the software was not able to accurately reproduce the visco-elasto-plastic behavior of such a complex fluid. However, in a macroscopic point of view, the physical-chemical interactions between the bubbles can be related to some non-Newtonians models, Bingham, Power Law, etc.

Therefore, an isothermal and laminar Bingham fluid will try to represent the conduct of an aqueous foam with a void fraction of 70% (70/100 air volume) and a bubble diameter of 0,5 mm. Table (1) shows the rheological parameters that describe each Bingham fluid that better approaches each experimental case (half-sudden expansion and fence).

RESULTS AND DISCUSSIONS

The results obtained from the simulations are presented in terms of streamlines, velocity profiles and pressure gradients for the two singularities studied (half-sudden expansion and fence). These ones will be compared against the experimental results of Aloui et al. [4, 5].

Streamlines

The streamlines for the fence and the half sudden expansion were calculated from the mean velocity fields from the CFD simulations (Figure 3). They show the creation of death zones for both geometries. The presence of the singularities causes the

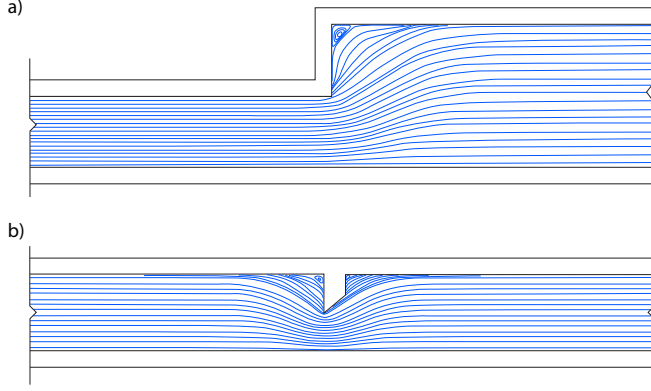


FIGURE 3: STREAMLINES OF THE MEAN VELOCITY FIELDS: a) HALF-SUDDEN EXPANSION b) FENCE

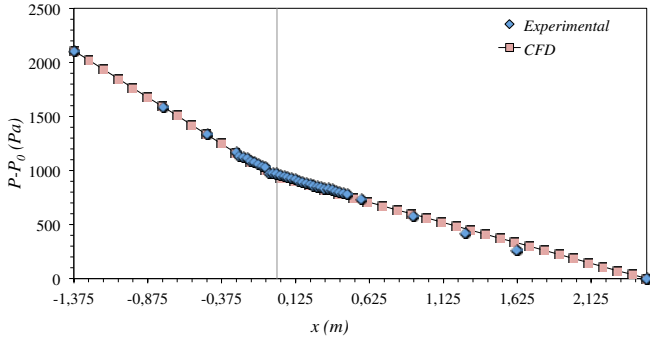


FIGURE 4: EVOLUTION OF THE STATIC PRESSURE FOR THE HALF-SUDDEN EXPANSION

formation in the immediate vicinity of both cases. In these regions the flow moves slowly and can even stagnate there. The same phenomena was observed from the experimental results. For this cases foam flow can be divided in two regions: the principal foam flow and the dead zones where the bubbles are almost motionless.

Pressure distribution

The pressure distribution is an essential element for modelling any kind of flow. Therefore, it is important to find out if our numerical simulations are able to recreate this behavior. Figs. 4 and 5 show the regular pressure evolution along the duct containing the half-sudden expansion and the fence for both experimental and numerical data. They have the same comportment, far from the singularities the static pressure varies linearly at the upstream and downstream of the duct.

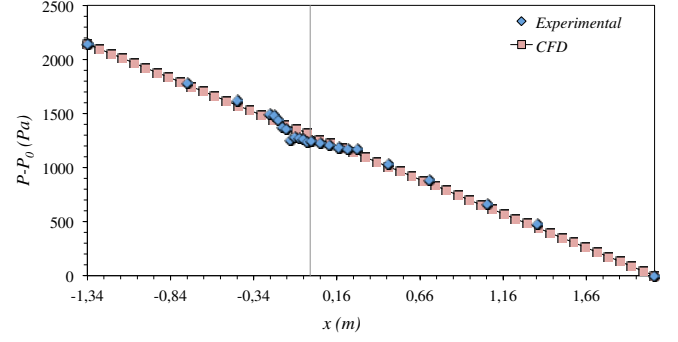


FIGURE 5: EVOLUTION OF THE STATIC PRESSURE FOR THE FENCE

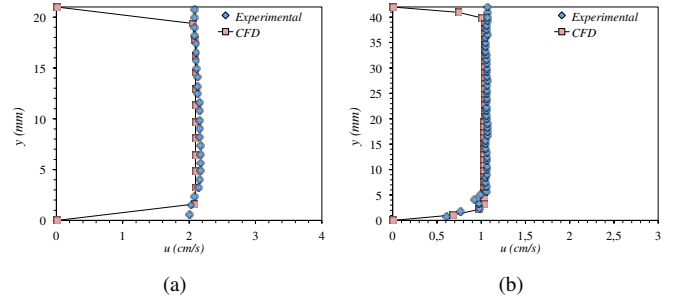


FIGURE 6: PROFILE OF THE AXIAL VELOCITY COMPONENT FOR THE HALF SUDDEN EXPANSION: (a) AT THE UPSTREAM AND (b) DOWNSTREAM

Local averaged velocity profiles

In order to realize the nature of the foam flow, upstream and downstream of the singularities, we have compared the axial velocity component profiles. Figs. 6 and 7 show the profiles for the half-sudden expansion and the fence respectively. We notice that the axial velocity presents the one-dimensional regime explained before. For the numerical results, the flow tends to shear more near the wall and the non-slipping condition is observed. However, all profiles approach considerable close.

Figs. 8 and 9 represent the comparison of some profiles of the axial velocity component along the lateral side of the duct, downstream of the expansion and both downstream and upstream of the fence. For the half-sudden expansion we note that the flow slows down as we move away from the singularity until it reaches a balance value (1.1 cm/s) and it remains constant until the exit of the duct. As for the fence, we remark an acceleration as we move closer to the singularity reaching a value of 4.3 cm/s . Then the flow returns to the same velocity that it had before passing the obstacle (2.15 cm/s). The presence of the death zones, for both cases is clearly noticed near the singularities where the axial velocity tends to 0. The absence of a negative axial velocity

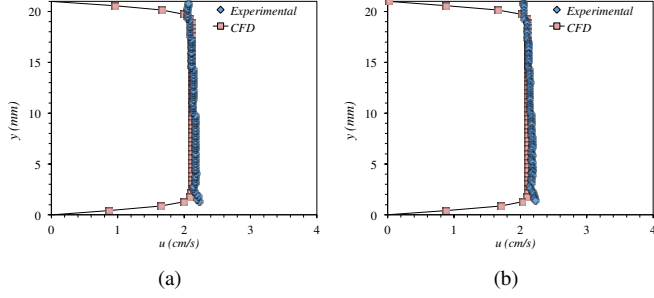


FIGURE 7: PROFILE OF THE AXIAL VELOCITY COMPONENT FOR THE HALF SUDDEN EXPANSION: (a) AT THE UPSTREAM AND (b) DOWNSTREAM

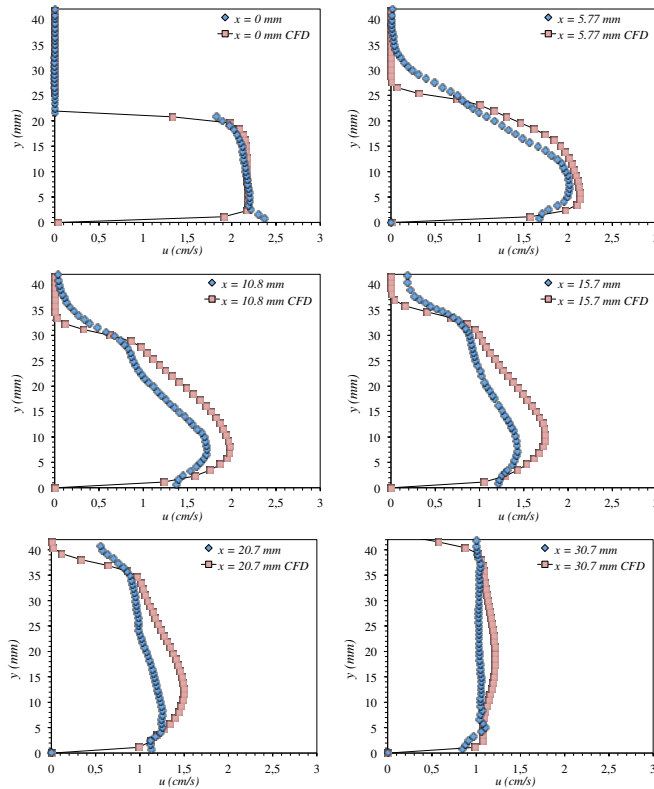


FIGURE 8: AXIAL VELOCITY PROFILE COMPONENTS AT DIFFERENT DISTANCES FOR THE HALF-SUDDEN EXPANSION

component indicates that there is no flow return in the opposite direction. The numerical results present a better approach for the half-sudden expansion case than for the fence one.

For the same plane the vertical velocity component profiles were obtained (Figs. 10 and 11). In the half-sudden expansion we notice that there are some vertical velocities that are more important in the immediate vicinity of the singularity. They are

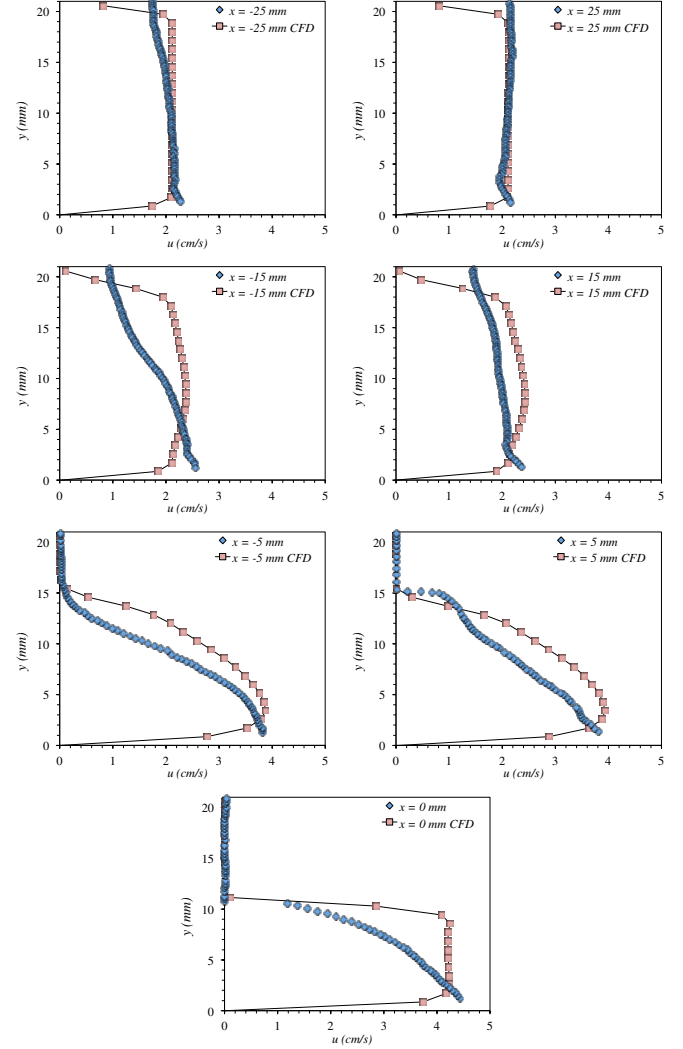


FIGURE 9: AXIAL VELOCITY PROFILE COMPONENTS AT DIFFERENT DISTANCES FOR THE FENCE

related to the upward movement of the flow trying to occupy the rest of the duct. These velocities also show that after the section expansion a fully developed flow is reached when moving away from the singularity. For the fence, as in a nozzle, the reduction of the section creates a descending vertical movement towards the passage section below the singularity. The foam leaves the obstacle to occupy the rest of the duct while decelerating gradually until reaching a constant velocity and returning to the one-dimensional flow. The numerical results present the same behavior as the experimental ones. However, some particular points present high deviations, $x = 20.7 \text{ mm}$ for the half-sudden expansion and $x = -5 \text{ mm}$ for the fence.

Fig. 12 shows the velocity components evolutions along the x direction near the top side of the half-sudden expansion. In

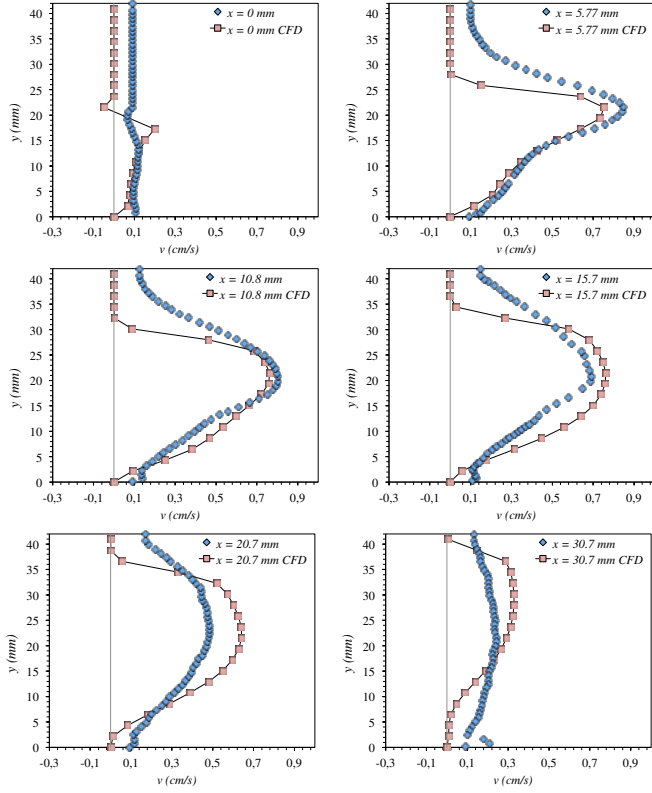


FIGURE 10: VERTICAL VELOCITY PROFILE COMPONENTS AT DIFFERENT DISTANCES FOR THE HALF-SUDDEN EXPANSION

the axial component graph we can clearly observe that the death zone, are with a zero velocity, for the numerical results has a size two times bigger than the experimental one. This means that the foam flow adapts faster than the Bingham fluid to the section change and it quickly starts to fill the void created by the singularity. However, they both seem to reach a fully developed flow near the same velocity (0.98 cm/s) and both curves behave equally. For the vertical component the same phenomenon is witnessed. The death zone prolongs farther away for the CFD simulations and then flow acceleration occurs filling the empty space at the top of the expansion. A maximum velocity is achieved; this one is bigger for the experimental results. The highest deviation between the experimental and the numerical results is 24%.

As for the fence Fig. 13 shows the longitudinal profiles in the vicinity of the singularity. The first one represents the axial velocity. We can observe that both, foam flow and Bingham fluid, slow down gradually upstream of the obstacle. Later on they stop completely at the bottom of the corner formed by the fence and the duct ($-30 \text{ mm} < x < 0 \text{ mm}$). In the downstream of the fence, the flows start to accelerate continuously, coming out of the second death zone, to reach a constant velocity, with a value close

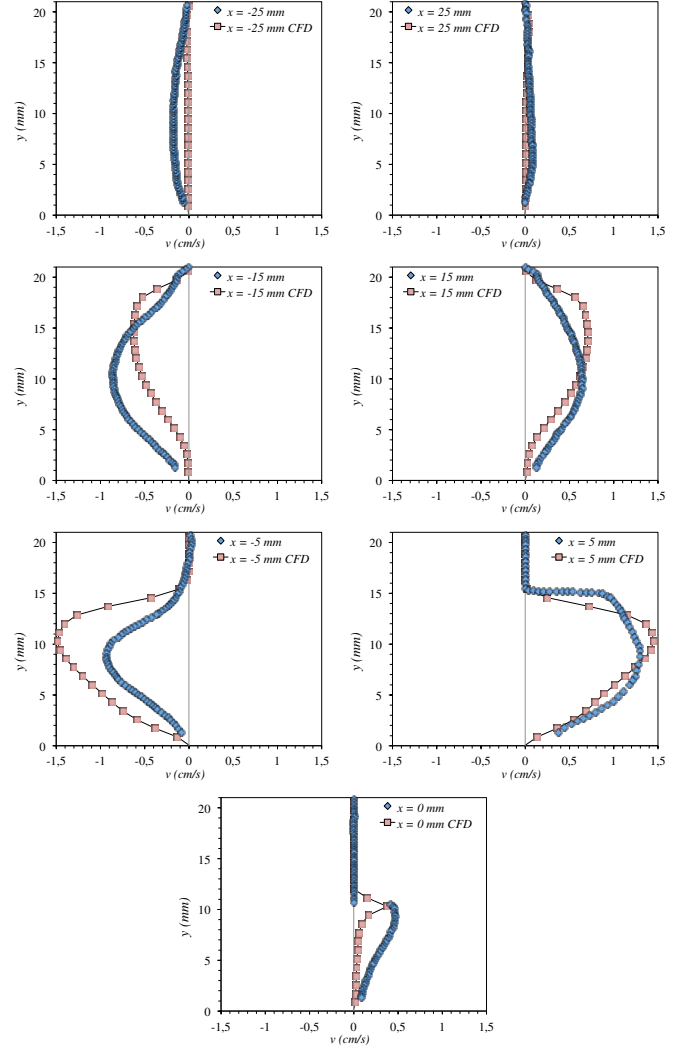


FIGURE 11: VERTICAL VELOCITY PROFILE COMPONENTS AT DIFFERENT DISTANCES FOR THE FENCE

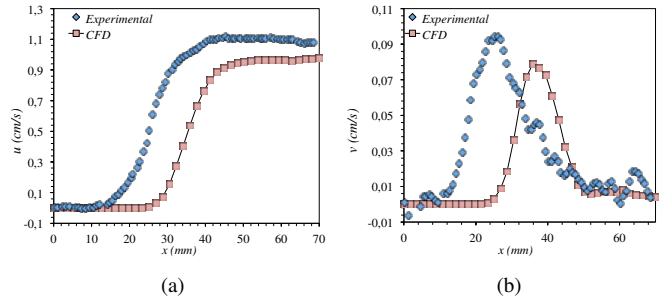


FIGURE 12: (a) AXIAL AND (b) VERTICAL VELOCITY EVOLUTIONS ALONG THE X-DIRECTION NEAR THE TOP SIDE DOWNSTREAM THE HALF-SUDDEN EXPANSION

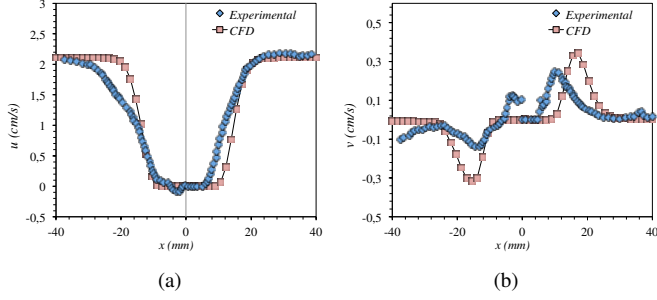


FIGURE 13: (a) AXIAL AND (b) VERTICAL VELOCITY EVOLUTIONS ALONG THE X-DIRECTION NEAR THE TOP SIDE IN THE VICINITY OF THE FENCE

to the inlet velocity (2.1 cm/s), far from the singularity. The second one plots the vertical evolution along the top side of the duct. Both results, numerical and experimental, show the acceleration and deceleration for the upstream and downstream respectively. The death zones at the both side of the fence can also be noted, more clearly for the numerical results. The deviation for the vertical profiles is much higher than for the axial ones, reaching a maximum difference of almost the double between the experimental and numerical data. The graphs appearance show once more that foam flow has a capacity to adapt to section change faster than the Bingham fluid.

Because these experimental results were only available for the half-sudden expansion, the variation of the axial and vertical velocity components along the expansion direction (x direction) are represented in Figs. 14 and 15 for different positions on the y direction. From the axial velocity profiles, we observe that the velocity all along the height of the channel descends until reaching the same value creating the *piston flow*. This behavior is expected for this kind of flow. This deceleration is due to the increase of the flow area section, after this change the flow tries to reach an equilibrium point and a minimal uniform speed far from the singularity. The same situation is equally described by the vertical profiles. We see the flow vertical acceleration filling the void after the expansion and then a transition period until the piston flow is achieved and the vertical velocity is null. Both numerical and experimental plots present the same appearance, and the maximum deviation between them is 16%.

CONCLUSIONS

In this paper, CFD simulations of Bingham fluids through a half-sudden expansion and a fence were compared against experimental results of a foam flow in the same conditions. The parameters choose to undertake this comparison were the flow streamlines, the pressure distribution along the whole channel length and the velocity profiles near the two singularities. The transport characterization of foam flow raises at least two prob-

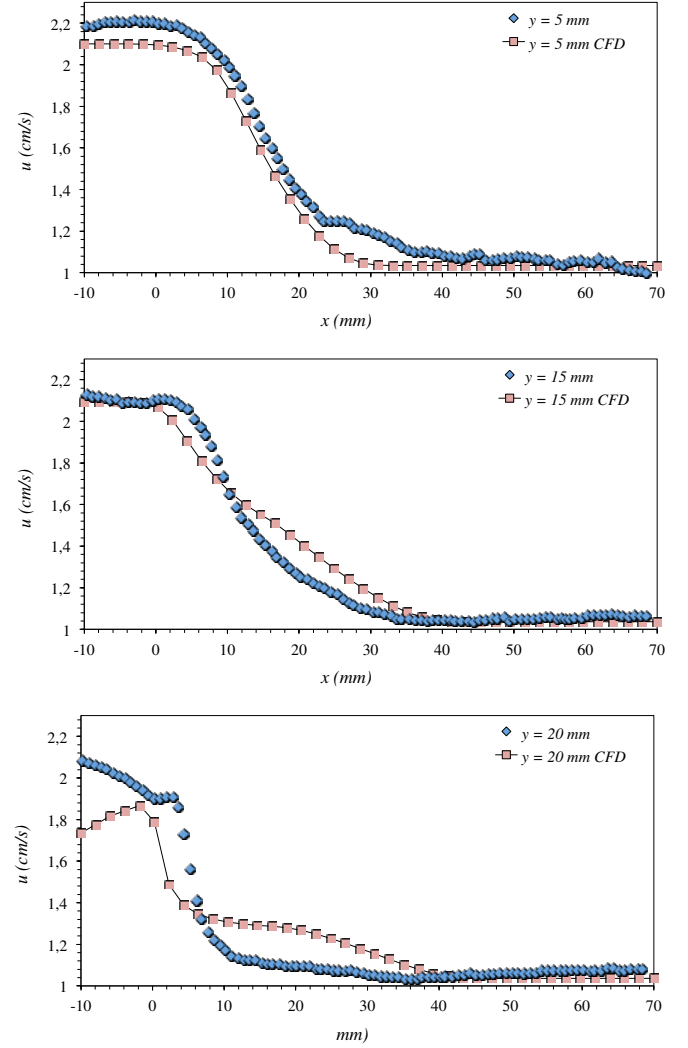


FIGURE 14: AXIAL VELOCITY IN THE VICINITY OF THE HALF-SUDDEN EXPANSION AT DIFFERENT HEIGHTS

lems: a stability problem and a rheology problem. To be able to simplify the rheological problem by relating such a complex fluid to a simpler one is an important step for simulations of foam flow and behavior comprehension. Both the CFD and the experimental results graphics present the same appearance when compared one against the other. The fluids, foam flow and Bingham, behave equally when faced against a half-sudden expansion and a fence. One of the most important aspect of any rheological model is the interaction between the wall and the fluid, wall shear stress, velocity gradient and pressure losses. The pressure distribution for the both geometries is exactly the same, with a maximum deviation of 5%. As for the velocities profiles, the slipping condition that bubbles have over the channel walls plays a major role in the difference between the numerical and the ex-

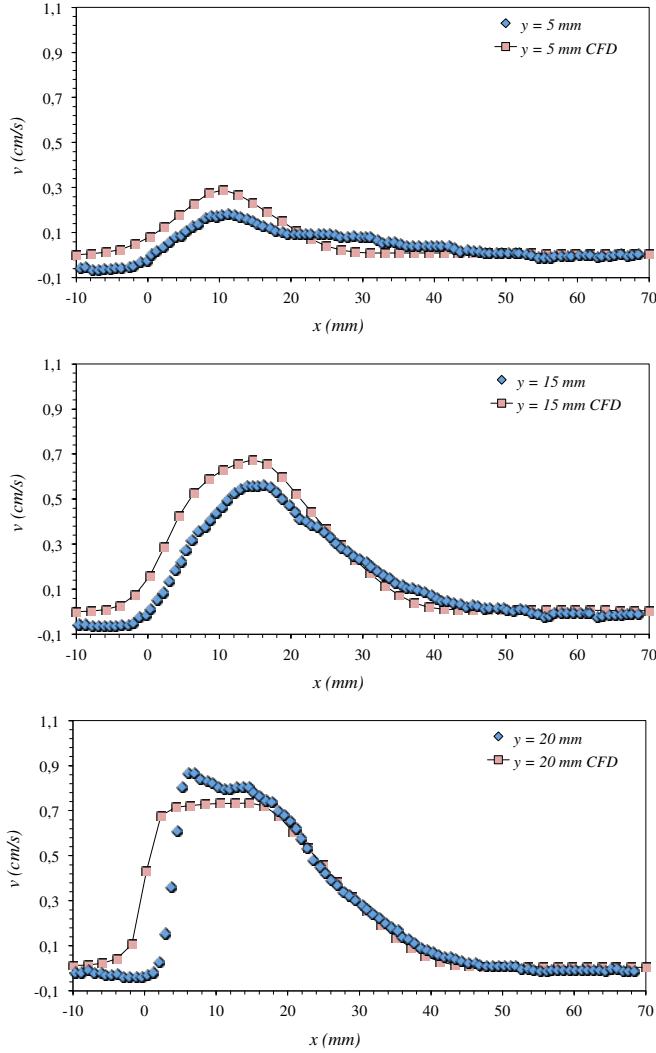


FIGURE 15: VERTICAL VELOCITY IN THE VICINITY OF THE HALF-SUDDEN EXPANSION AT DIFFERENT HEIGHTS

perimental curves. Nevertheless, the Bingham fluids approach very closely to the real behavior and most of the graphics present a deviation of 10% but in some cases it can go over to 25% and even 50%. These results show that a Bingham fluid can accurately predict the behavior of a foam flow, for a slow velocity, when faced against different geometries. Being able to reduce to essentials a complicated flow facilitates future simulations of foam flows. Further studies should be focus in closing the gap obtained for the velocity profiles; therefore, obtaining an exact similarity between a rheological model and the real foam flow. In order to do this, 3D simulation should be undertaken for this same conditions and the use of more elaborated models is recommended, such as the Power Law rheological model and even

a visco-elasto-plastic model.

REFERENCES

- [1] Cantat, I., Cohen-Addad, S., Elias, F., Granier, F., Hhler, R., Pitois, O., Rouyer, F., and Saint-Jalmes, A., 2013. *Foams: Structures and Dynamics*. Oxford.
- [2] Cohen-Addad, S., Hoballah, H., and Hhler, R. "Viscoelastic response of a corsesing foam". *Phys. Rev.*, **57**.
- [3] Weaire, D. "The rheology of foam". *Curr. Opin. Coll. Interface Sci.*, **13**.
- [4] Aloui, F., and Madani, S. "Wet foam flow under a fence located in the middle of a horizontal duct of square section". *Colloid. and Surface. A*, **309**.
- [5] Aloui, F., and Madani, S. "Experimental investigation of a wet foam flow through a horizontal sudden expansion". *Exp. Therm. Fluid Sci.*, **32**.
- [6] Chovet, R., Aloui, F., and Labraga, L. "Wet foam flow in horizontal square duct and through singularities". *FEDSM 2013*.
- [7] Herzhaft, B., Kakadjian, S., and Moan, M. "Measurement and modeling of the flow behavior of aqueous foams using a recirculation pipe rheometer". *Proc. of 5th European Conference of Foams, Emulsions and Applications*, **263**.
- [8] Feitosa, K., and Durian, D. "Gas and liquid transport in steady-state aqueous foam". *Eur Phys J*, **26**.
- [9] Ireland, P. "Coalescence in steady-state rising foam". *Chem. Eng. Sci.*, **64**.
- [10] Gabrielaitiene, I., and Gyls, J. "Modelling of aqueous foam flow for developing energy-efficient heat exchanger applications". *The 8th International Conference of Environmental Engineering*.
- [11] Blondin, E., 1999. "Etude experimentale d'une mousse humide en coulement en conduite horizontale de section rectangulaire". PhD Thesis, Universit de Nantes, Nantes, France.