NUMERICAL INVESTIGATION AND ANALYSIS OF A DISPERSED TWO-PHASE FLOW ACROSS A SINGLE RIGID CYLINDER.

William Benguigui, Enrico Deri, Jerome Lavieville, Stephane Mimouni, Elisabeth Longatte
Electricite de France, Research&Development Division, Fluid Mechanics Energy and Environment Department
6 Quai Watier, Chatou 78401, France
IMSIA, UMR EDF-ENSTA-CNRS-CEA 9219
Paris-Saclay university, 91762, Palaiseau, France
Email: william.benguigui@edf.fr

ABSTRACT
The numerical simulation of interaction between cylindrical structures and two-phase flows is a major concern for industrial applications, especially where cross-flows may cause damages, in heat exchangers for example. In order to understand the phenomenon of vibrations induced by two-phase flows, many experiments were conducted with reduced-scale models using several simulant two-phase mixtures. In single-phase flow, the phenomenon has been characterized with dimensionless numbers. In the present work, the authors are interested in having the simplest configuration to study the influence of inlet parameters and fluid properties and consequently of two-phase flow dimensionless numbers. In 1986, Inoue et al. [1] pointed out different properties of a dispersed air/water flow around a single rigid cylinder for various inlet void fractions, velocities, and bubble to cylinder size ratios. Based on this study, we present a numerical analysis with a finite-volume CFD software dedicated to multi-phase flow. The two-fluid approach provides a satisfactory agreement between Inoue’s experiments and simulations. A numerical investigation is then proposed on the mixture influence. The sensitivity to surface tension, which might drastically change the vortex-shedding, is studied. Finally, the authors try to characterize this specific configuration with dimensionless numbers.

INTRODUCTION
Flow-induced vibrations of tubes in two-phase flow heat exchangers are a concern for the nuclear industry since it might cause damages. Consequently, when operating conditions are too severe to allow convenient measurement procedures, relevant phenomena are investigated by means of reduced-scale experiments using modeling fluids which are different from the real ones. Similarities have then to be taken into account. Some studies tried to address this topics in the past. Gay et al. [2] compared air/water with freon/water mixtures in a triangular pitch bundle. Axisa et al. [3] compared steam/water with air/water. Benguigui et al. [4] compared water/air and freon/freon with numerical simulations. Experimen-
tal studies involved large facilities and are often limited to the analysis of global parameters.

Hulin et al. [5] studied vortex emissions behind bluff obstacles in a gas-liquid vertical flow. Inoue et al. [1] studied bubbly flow around a single cylinder for different velocities, void fractions and size ratios. Meng et al. [6] studied the discrepancies between air-water and steam/water for a cross flow around a single cylinder for low void fractions. More recently, Murai et al. carried out experiments on different kinds of obstacles, mainly with dispersed bubbles rising in a fluid at rest. Pascal-Ribot & Blanchet [9] measured forces on a single cylinder under different two-phase flow regimes. Zhihua et al. [8] studied the two-phase flow regime transition for an air/water upward flow around a single cylinder. In the present document, based on the experiment from Inoue, the authors focus on dispersed flow around a single cylinder in order to enrich the conclusion from the previous cited studies thanks to numerical simulation.

Two-phase flow modeling used for the study is shortly presented [10]. After a presentation of the experiment, a mesh refinement study is performed. Then, the influence of inlet void fraction and inlet velocity is considered. Finally, surface tension being one of the parameters defining a mixture, its influence is also numerically investigated.

**NUMERICAL MODEL**

In the present work, numerical simulations are performed with NEPTUNE_CFD software [10], a CFD code dedicated to multiphase flows, based on the two-fluid (extended to \( n \)) approach. A finite-volume discretization is used with a collocated arrangement for all variables. Using a pressure correction approach [12], it is able to simulate multiphase flows by solving a set of three balance equations for each field. These balance equations are deduced from volumetric averaging of local instantaneous balance equations. Fields can represent different kinds of multiphase flows. However, the present work focuses on liquid-gas flows only and restricted to adiabatic cases, simplifying the system to the mass and momentum balance equations for each phase \( k \):

\[
\frac{\partial (\alpha_k \rho_k)}{\partial t} + \vec{\nabla} \cdot (\alpha_k \rho_k \vec{U}_k) = 0 \\
\frac{\partial (\alpha_k \rho_k \vec{U}_k)}{\partial t} + \vec{\nabla} \cdot (\alpha_k \rho_k \vec{U}_k \vec{U}_k) = -\alpha_k \vec{V} + \alpha_k \rho_k g \\
+ \vec{\nabla} \cdot \vec{\tau}_k + \sum_{p=1}^{N} M_{p \rightarrow k}^{\text{hydro}} \\
\]

where \( \alpha, \rho, \vec{U}, \Gamma, P, \tau, \vec{g} \) and \( M \) are respectively the volume fraction, the density, the velocity, the mass transfer, the pressure, the Reynolds-stress tensor, the gravity and the interfacial momentum transfers to the phase \( k \).

For the present study, gas and liquid are respectively modeled with a continuous and a dispersed field. The closure laws for the dispersed field are the momentum interfacial transfer between phases: the classical drag of Ishii [13] developed for spherical and slightly deformed bubbles, the lift of Tomiyama [14] postulated for the different bubble shapes, the virtual-mass of Zuber [15] being the force induced by the fluid displaced by bubbles; and turbulent dispersion [16] expressing the interaction between bubbles and turbulence.

\[
M_{p \rightarrow k}^{\text{hydro}} = M_{p \rightarrow k}^{D} + M_{p \rightarrow k}^{L} + M_{p \rightarrow k}^{AM} + M_{p \rightarrow k}^{TD} \\
\]

A second order turbulent model dedicated to high-Reynolds two-phase flow is used; further details can be found in [18]. The bubble size distribution modeling has been developed for bubbly flow based on the moment density method [19].

**DISPERSED WAKE AROUND A CYLINDER**

**Experiment**

The experiment from Inoue et al. [1] deals with two-phase flow around a confined single rigid cylinder. It is a vertical upward air/water flow. Air is injected into water in order to have a uniform bubbly two-phase flow. The cross section is a rectangle of 120x60 mm. The total length of the channel is 1900 mm and the cylinder is located at 1300 mm from the inlet. The length of the cylinder is 58 mm and its diameter is between 10 and 40 mm. The flow characteristics are investigated for different bubble to cylinder diameter ratios, inlet velocities and inlet void fractions. The main parameters measured are the averaged void fraction, liquid velocity and static pressure.
FIGURE 1: Photographs of the wake around a cylinder with $\alpha_{\text{inlet}} = 8\%$, $U_{\text{inlet}} = 0.45, 0.9, 1.9 \text{ m/s}$ and $D = 40 \text{ mm}$ [1].

The most noticeable effect is local void fraction peak appearing in the vicinity of the cylinder near the separation point and in the wake behind the cylinder. By looking at the different cases, different observation are reported. Void fraction is higher at the separation point and at the wake of the cylinder. Due to the static pressure gradient, at the front and rear of the cylinder, there is a liquid layer where bubbles can hardly penetrate. At the rear, in contrast with the front, its thickness is reduced when the velocity is increased. When the mean velocity is increased, the peak of void fraction is also larger. The densification of bubbles is bigger when the cylinder diameter is increased. However, void fraction distribution is not highly affected by the increase of void fraction from 4% to 16%.

Validation of numerical results

Since the software is able to generate a uniform inlet bubble distribution, the height of the domain is reduced from 1900 mm to 1000 mm with a cylinder located at 400 mm. The case with $\alpha_{\text{inlet}} = 8\%$, $U_{\text{inlet}} = 0.9 \text{ m/s}$ and $D = 40 \text{ mm}$ is used for the presented validation. The numerical model presented previously is used. Top and bottom surfaces are respectively identified as inlet and outlet. The other walls have a non-slip condition. The simulation is unsteadily performed during 5 seconds. A mesh refinement study is performed and compared to experimental results. In the present case, the Reynolds number is 36 000 which is really high. The mesh has thereafter to be very fine close to the cylinder wall to capture correctly the vortex shedding, the finest mesh is still running. However, the result obtained with 3 mesh refinements is presented in Fig.2. In comparison with the experiment, the more the mesh is refined, the closer we are from the experiment. The numerical prediction is not accurate where the vortices are created. In this area, the bubbles are captured by the vortices which explains the densification of bubble. But in the simulation, the vortices are not accurately reproduced given that the Reynolds number is extremely high. Consequently, at this location, the simulation is not accurate even if the behavior is correctly reproduced. For the rest of the study, the most refined mesh available at this time is used to enrich the experimental conclusions.

FIGURE 2: Averaged void fraction distribution around a cylinder with $\alpha_{\text{inlet}} = 8\%$, $U_{\text{inlet}} = 0.9 \text{m/s}$ and $D = 40 \text{ mm}$. From left to right, there are experimental data, mesh 1, mesh 2 and mesh 3.

Influence of inlet void fraction

According to Fig.3, the inlet void fraction influences the location of the vortices. In fact, the lower is the void fraction, the closer the vortices are from the cylinder. By increasing the void fraction, the distribution is similar but larger. If the two cases with 4% and 8% have similar profiles, the last one is different. This is due to the bubbles affecting the liquid flow. This observation was also made by Pascal-Ribot et al. [9] in another way. They showed that under 11% of void, the forces were similar. Between 11% and 20% however, the force spectra exhibits a clear dependence on void fraction: it increases with the void.
Influence of inlet velocity

According to Fig.2-4, the averaged void fraction profiles depend on the inlet velocity. For $U_{inlet} = 0.9, 1.9 \text{ m/s}$, bubbles are captured by the vortices created by the liquid flow around the cylinder. The higher is the velocity, the more concentrated and less dissipated is the void fraction distribution. However, for lower velocity, the profile is strongly different. Thanks to numerical simulation, other quantities are available such as the diameter or the gas and liquid velocities. Since the velocity is too high, the bubble coalescence is low and the diameter remains constant. However, for low velocity, the bubble velocity is driven by the buoyancy, consequently the coalescence is more present and the bubble diameter is higher in the rear of the cylinder. This explains the discrepancies between liquid and gas velocity profiles in Fig.5.
Surface tension influence

Inlet void fraction and velocity (or size ratio) are presented in Inoue’s article and in the present work. Numerical simulations are important to investigate this kind of flow, since once it is validated, many variables are available such as velocity fields, diameters, turbulence and many others. In previous cases, only flow characteristics are changed, e.g. the Reynolds number. In the present work, in order to show the important influence of the mixture, we changed the surface tension of the mixture to see how it might modify the flow. In fact, surface tension is an important parameter which is present in bubble characteristic non-dimensional numbers like Eotvos, Morton or Weber:

\[
Eo = \frac{\Delta \rho g D^2}{\sigma}, \quad Mo = \frac{g \mu L^4 \Delta \rho}{\rho L^3 \sigma^3}, \quad We = \frac{\rho U D}{\sigma}
\] (3)

Moreover, the surface tension is related to the bubble diameter given that it is the liquid-gas interface strength. Consequently, the lower is the surface tension, the lower is the bubble diameter.

Based on the same initial case, we only changed the surface tension to study its impact on void fraction distribution. According to the void fraction profiles on Fig.6, by lowering is the surface tension, the vortex shedding is less disturbed and close to the single-phase flow case since bubbles are smaller. The surface tension has consequently an important influence on turbulent energy which is of primary interest in two-phase cross flow given that it is responsible of vibrations.

CONCLUSION

In the present work, an overview of literature is provided on two-phase flow around a single rigid cylinder. After presenting the numerical modeling of two-phase flow, a mesh refinement study is performed on one case. Due to the high Reynolds number, the mesh is unfortunately not refined enough. Consequently, as the numerical result is considered sufficiently in agreement with the experiment, the most refined mesh is used to investigate the influence of inlet void fraction and velocity. The inlet void fraction from 4% to 16% does not change significantly the void fraction distribution even if the densification of bubble and the liquid layer behind the cylinder are longer with a higher void fraction. Since bubbles remain spherical and with a constant diameter, the flow is not drastically changed. However, for higher void fractions, coalescence might appear and create bigger gas structures which would significantly modify the flow. The inlet velocity from 0.45 m/s to 1.9 m/s has an important impact on the flow. Due to the presence of bubbles, the liquid flow is disturbed, especially for low velocities. In fact, when the bubble velocity due to buoyancy is higher than the liquid velocity, the liquid flow is disturbed by bubbles. However, in comparison, for a higher liquid velocity, bubbles are following the liquid flow and captured by the liquid vortices. The surface tension is of primary interest since it is strongly related to turbulence. By lowering the surface tension, bubble are smaller and disturb the liquid flow less. In comparison, by increasing the surface tension, bubbles are larger, and dissipate the turbulent vortices.

This study is a first step to characterize this kind of flow. By changing physical properties of mixtures, it is possible to quantify their influence, consequently the discrepancies between mixtures. In the future, mass and viscosity ratios will be studied in details to complete this work.
ACKNOWLEDGEMENTS

Authors want to thank both EDF R&D projects: Qual-IFS-GV, devoted to the assessment of SG tube vibration risk; and NEPTUNE funded by EDF (Electricité de France), CEA (Commissariat l’Energie Atomique et aux Energies Alternatives), Framatome and IRSN (Institut de Radioprotection et de Surete Nucleaire).

REFERENCES


