Obtaining of Analytical Relations for Hydraulic Parameters of Channels With Two Phase Flow Using Open CFD Toolbox

E Varseev
ROSATOM CICE&T, Obninsk 249039, Kurchatova 21, Russia

Email: eugeny.varseev@gmail.com

Abstract. The present work is dedicated to verification of numerical model in standard solver of open-source CFD code OpenFOAM for two-phase flow simulation and to determination of so-called “baseline” model parameters. Investigation of heterogeneous coolant flow parameters, which leads to abnormal friction increase of channel in two-phase adiabatic “water-gas” flows with low void fractions, presented.

1. Introduction

Two- and multiphase flows with bubble structure is widely presented in traditional and nuclear power industry, in chemical and technological production, pipeline transport of hydrocarbons, metallurgical and other equipment. Regimes of normal operation for such systems are justified by experimental and numerical methods based on regulatory and reference data. However transient and abnormal regimes, especially in power industry are investigated by the one-dimensional semi-empirical and empirical approaches due to many possible scenarios of external effects and critical phenomena impact, such as heat flux crisis, for example. This limits recommendations developed to the specific geometry and conditions range, and this, in turn, leads to usage of overvalued coefficients and reinsured designs.

During the last decades, a large amount of experimental data on heterogeneous distributions of disperse phase local parameters (gas fraction or void fraction in particular) and accompanying phenomena of abnormal increase of flow integral characteristics – friction, heat- and mass transfer, for example [1]. The models presented in the majority of the one-dimensional codes, not always describe gas distribution via radius of the channel adequate enough and don’t take into account abnormal growth of friction at the wall of the channel [2], and for this reason they are in the process of active verification. Implementation of CFD codes for multiphase flow simulation with “Euler-Euler” approach allows to overcome shortcoming of the one-dimensional semi-empirical codes.

The present work is dedicated to verification of numerical model in standard solver of open-source CFD code OpenFOAM for two-phase flow simulation – twoPhaseEulerFoam, and to determination of the “baseline” model parameters. The investigation of heterogeneous distributions of non-equilibrium coolant flow, which leads to abnormal friction increase of the channel in two-phase adiabatic “water-gas” flows with low void fractions, presented.

There are a significant number of works in open literature, dedicated to two-phase flow modeling with CFD codes [4-7] and all of them have the following distinctive features:
- authors use modified solvers with two-phase models;
- for verification they use international databases for radial void fraction distribution;
authors basically don’t consider effects of abnormal friction increase.

On contrary, during this work authors used default OpenFOAM solver for velocities and void-fraction profiles calculation; the results of simulation were compared against Russian experimental results including comparison of friction at the wall. As a result the friction coefficient of channels walls were obtained in the wide range of gas flow ratio β.

In a result of series of simulations the data on upward two-phase flow with abnormal friction conditions were obtained and analyzed. Investigation of heterogeneous void fraction distributions of two-phase adiabatic “water-gas” flow is presented. Simulation results using OpenFOAM code demonstrate abnormal growth of shear stress in the channel, which was observed in the experiment earlier.

The problem description and simulation

Experiments described in work [1] were simulated using OpenFOAM code. Test section of the experimental facility is a round channel (diameter d = 86.4 mm; height h = 6.75 m) through which two-phase flow “water-gas” with fixed flowrate and void-fraction was pumped from the bottom towards the upper end of the channel. In the result of the series of experimental runs the distribution of void fraction, velocities of each phase via radius of the channel and shear stress on the wall of the channel were measured.

Simulation domain is a wedge mesh (5˚) with inlet, outlet, wall and wedge-type boundaries (fig. 1a). The mesh was created using default blockMesh utility, incorporated in OpenFOAM.

As a first thing, the one-phase flow simulations were carried out with water velocity at the inlet Ul = 0.79 m/s for different meshes – ~2000, ~4000 and ~10 000 cells; comparison of experimental and simulated velocity profiles was performed. Based on the simulation results the decision was made that cells number increase above 4000 is not useful, since the maximum discrepancy of experimental and simulated data is 5% and three was no pressure loss change. On the next step the ten regimes of two-phase flow were simulated, different in velocity of phases and mean void fraction at the inlet.

In twoPhaseEulerFoam solver algorithm five interfacial models are implemented: virtual mass, drag and lift forces, turbulent dispersion and wall lubrication. There are from three to five different variants of each force formulation in twoPhaseEulerFoam solver in OpenFOAM. In the simulations the types of interfacial models formulation and variants of their constant, taken from the widely applied in literature “baseline” model for CFD simulation of similar two-phase flow cases were used. Its main parameters presented in the table.

The simulation data were compared against the experimental one – see fig. 1b. In the course of simulation at the height of the channel h = 0.3..0.5 m from the inlet and after approximately 7 s of simulation time the void fraction with typical saddle-shape was developed. This shape is typical for the gas flow ratios in the 0.05…0.25 range, the further increase of β gives friction of the channel in accordance with Armand relation [8] – see fig. 1c.

On the fig. 1b the profile of bubbles accumulated near the wall of the channel are shown. This increase of gas phase concentration near the wall (saddle-shaped profile) is considered to be the reason of abnormal friction on the wall compared to the friction described by homogeneous flow model.
Fig. 1. Simulation domain (a); distribution of void fraction (OpenFOAM results) vs radius of the channel (b) and dependence of relative shear stresses on gas flow ratios (c).

To ensure obtaining robust results several preliminary steps were performed beforehand. First, the mesh independency check was performed: simulation of one-phase flow with the same velocity at the inlet and diameter of the channel was done using different meshes – see Fig. 2. The next step was comparing velocity profile against experimental at $\alpha = 0.05$ – see Fig. 3.

Stabilization process of void fraction profile is occurring along the channel height. One-third of the channel height is necessary for profile to take the typical saddle shape form. – see Fig. 4.

--- mesh 1, — mesh 2; —— mesh 3; • – experimental data [1]'

Fig. 2. Determination of optimal mesh size, one-phase flow.
Implementation of default twoPhaseEulerFoam solver in more recent OpenFOAM 3.0 for simulation allowed to consider new effects – non-uniform bubbles diameter and effect of bubbles on turbulent parameter (Sato turbulence model) – see Fig. 5. The latter fact resulted in decreased turbulent viscosity – see Fig. 6. This fact was also observed by Japanese researchers [9].
Fig. 5. Simulation results using OpenFOAM 3.0 ($kOmegaSST$-Sato, $d_b=0.5\div1.5 \text{ mm}$).

Fig. 6. Turbulence suppression in two-phase flow.

**«Baseline» model parameters used in simulation**

| Name                        | Value           | Dimension |
|-----------------------------|-----------------|-----------|
| Bubble diameter             | $d = 10^{-3}$   | m         |
| Surface tension             | $\sigma = 0.072$ | J/m$^2$   |
| Virtual mass coefficients   | $C_{VM} = 0.5$  |           |
| Lift force coefficients     | $C_L = 0.9$     |           |
| Wall lubrication coefficients | $C_{WL1} = -0.01$; $C_{WL2} = -0.05$ |           |
| Turbulent dispersion coefficients | $C_{TD} = 0.5$    |           |
| Drag force coefficients     | $C_D = 0.5$     |           |

**Discussion and conclusion**

In the work the result of numerical simulation of void fraction distribution via radius of the channel with two-phase flow with low ($\beta = 0.05\ldots0.2$) gas flow ratios are presented. In the area of the low void-fractions simulation results demonstrate abnormal increase (six times increase in the maximum) of shear stress in the round channel, which was observed experimentally [1]. Based on simulation
results the decision was made that OpenFOAM code is adequately simulates saddle-shaped void-fraction profile, which is believed to be the reason of abnormal increase of shear stress in channel.

In present time 2D/3D codes and OpenFOAM in particular are used to investigate different thermal hydraulic problems of two-phase non-equilibrium turbulent flows. But variety of physical effects accompanying dynamics processes of momentum, heat and mass transfer and complicated by generation of areas unstable in time, makes developing of approaches to the modeling of such phenomena difficult and developing recommendation for their validation problematic. This uncertainty simulated publications [3-7] on the listing of physical models available for selection of a “baseline” model [3]. Block-scheme of the «baseline» is shown on Fig. 7.

Fig. 7. Block scheme of iterative development of «baseline» model for two-phase flow simulation.

The most complete description of the two-phase flow fields in Eulerian representation taking in consideration interfacial interaction of phases in the form of 3D four-field model can be obtained by statistic averaging of local parameters in time [5]. This model is formulated in a system of conservation laws governing the mass, momentum and energy balance for each phase. But since averaged fields are interconnected, the effects of phase’s interaction are used in the model by source terms in the right part of the conservation laws. Such four-field model serves for generalization and extension of two-phase flow regimes simulation: from disperse bubbles in subcooled boiling phenomena to disperse drops of water in overheated steam.

These 3D conservation laws for multiphase media is a general structure for interconnection of momentum, heat and mass transfer exchange and demand a lot of additional information for their closure in regard of variables we are looking for. Reviews and recommendations for of such model, including all the necessary relations, are reported in [10, 11], and their connection with 1D best-estimate codes is presented in [2].

The iterative approach to improvement of two-phase flow simulation model incorporated in OpenFOAM code is presented in this work.

References
[1] Nakoryakov V.E. et al. Local Characteristics of Upward Gas-Liquid Flows // Int. J. Multiphase Flow – 1981. – V. 7. – P. 63-81.
[2] Kornienko Yu.N. Generalization of analytical integral forms of friction, heat and mass transfer coefficients for non-equilibrium two-phase flows. Annular channels and fuel assemblies. //
Issues of atomic energy. Physics of nuclear reactors series, 2013. Issue.4. – P. 61-76. (in Russian)

[3] Lucas D., Rzehak R., Krepper E., et al. A strategy for the qualification of multi-fluid approaches for nuclear reactor safety // Nuclear Engineering and Design – (2015), http://dx.doi.org/10.1016/j.nucengdes. 2015.07.007

[4] Tripathi R., Buwa V. Numerical Simulations of Gas-Liquid Boiling Flows Using OpenFOAM. // IUTAM Symposium on Multiphase Flows with Phase Change: Challenges and Opportunities, Hyderabad, India (December 08–11, 2014).

[5] Muki R.V. Modeling of bubble coalescence and break-up in turbulent bubbly flow. // Int. J. of Multiphase Flow 62 (2014) 52–66.

[6] Colombo M., Fairweather M. Multiphase turbulence in bubbly flows: RANS simulations. // Int. J. of Multiphase Flow 77 (2015) 222–243.

[7] Lahey, R. T. On the direct simulation of two-phase flows / R. T. Lahey // Nuclear Engineering and Design. – 2009. – V. 239. – P. 867-879.

[8] Armand A.A., Nevstryaeva E.I. Investigation of mechanism of two-phase flow in vertical tube. // Izv. VTI Vol. 2, P. 1-8.

[9] Serizawa A., Kataoka I. Turbulence suppression in bubbly two-phase flow. // Nuclear Engineering and Design, Volume 122, Issue 1, 1990, Pages 1-16.

[10] Bestion D., Lucas D., Anglart H. // The 20th Int. Conference on Nuclear Engineering ICONE20-POWER2012. – Anaheim, California, USA, July 30–August 3, 2012.

[11] Kornienko Yu.N. Multiphase and multiscale models and codes of non-equilibrium thermal hydraulic for safety analysis of NPP. Short review of recent achievements. // Proceedings of “Thermal hydraulics 2012” conf. Obninsk, 24-26 October of 2012. – Obninsk: IPPE, 2013. – Vol. 1. – P. 81-99. (in Russian).