NUMERICAL APPROACHES TO SIMULATE TWO-PHASE FLOWS

SEBASTIAN ROGER

Student of Civil Engineering
Department of Hydraulic Engineering and Water Resources Management,
RWTH Aachen University, 52056 Aachen, Germany
(Tel: +49-241-80-97778, Fax: +49-241-80-92275, e-mail: roger@iww.rwth-aachen.de)

Abstract

Multiphase flow is the simultaneous flow of several phases covering a wide range of applications in technical and industrial processes (SOMMERFELD, 2000). Additionally, it is encountered in numerous natural phenomena of our environment. The disciplines of chemical and process engineering have gained wide experience associated with multiphase systems (KUIPERS & VAN SWAAIJ, 1998). But several environmental phenomena, as for example erosion, transport and sedimentation of solid material in water bodies as well as (artificial) aeration or cavitation in flows are in the focus of interest in hydraulic engineering and can be described by two-phase systems, too. However, the design of hydraulic structures (i.e. spillways, weirs, bottom outlets, cascades) was primarily based on empiricism related to two-phase flow effects. Meanwhile there are hydraulic engineers studying aspects of air-water flows like turbulence (CHANSON & TOOMBES, 2002) or the prediction of cavitation inception (FARRELL, 2003). This paper tries to make a contribution to this "two-phase approach" in hydraulic engineering by starting with a feasibility study.

Depending on the volume fraction of the involved phases and their state of aggregation, the character of the flow differs significantly. Thus, a classification scheme for two-phase flows is introduced applying the proposed criteria on dilute, dispersed two-phase flows (Fig. 1) without phase transition. The numerical computation of two-phase flows may be performed in different ways related to the specific physical properties in the flow field under consideration. Moreover it is part of new CFD-applications making the highest demands on computer resources. Simulations are performed on different levels of complexity related to the resolution of the interface between the phases and the turbulence modelling. As a direct numerical simulation of dispersed two-phase flows in the field of engineering problems is not yet practicable, two approaches based on the REYNOLDS-averaged NAVIER-STOKES equations are commonly applied, namely the EULER-EULER and the EULER-LAGRANGE method.

Two practical test cases, the particle dispersion in a single-sided backward-facing step flow and the bubbly flow over a bottom-fixed sill in a cavitation channel are examined (Fig. 2). The air-solid step-flow has a classical setup regarding geometry and boundary conditions that is often used to verify numerical approaches in fluid mechanics. The liquid-gas flow provides a completely different configuration additionally featuring the phenomenon of cavitation. Numerical simulations of the flow are performed with the EULER-LAGRANGE approach which is already implemented into the Finite Volume code of the commercial software package COMET (Version 2.00, ICCM GmbH, Hamburg). A comparison of the numerical results with experimental data and other computations show that COMET is able to accurately model dilute particulate flows, if the particle size is

calibrated. However the calculation of the bubbly flow does not yield satisfactory results because essential mechanisms like certain aspects of phase-interaction, two-phase turbulence and microscopic processes like particle-wall collisions, inter-particle collisions, fragmentation and agglomeration of solid particles as well as coalescence, growth or collapse of bubbles and droplets, are neglected. Due to the lack of detailed experiments, these mechanisms can not be simulated in their full complexity.

Numerical simulations of two-phase flows with the objective of accurately modelling phenomena, which are relevant in hydraulic engineering, deserve further study. Thus, proprietary developments which account for specific hydraulic questions are desired. In most instances computations of two-phase flows on a large scale are not yet practicable. Therefore computations should be performed rather regarding hydraulic structures than in terms of whole stream courses. In spite of the great necessity of enhancing the familiar solution techniques from single-phase to two-phase flows, the presence of an additional phase causes new effects, which are understood insufficiently to allow a derivation of a model from basic principles of physics. Hence, the provision and validation of appropriate models by means of extensive experimental investigations with sophisticated measurement techniques is eagerly required.

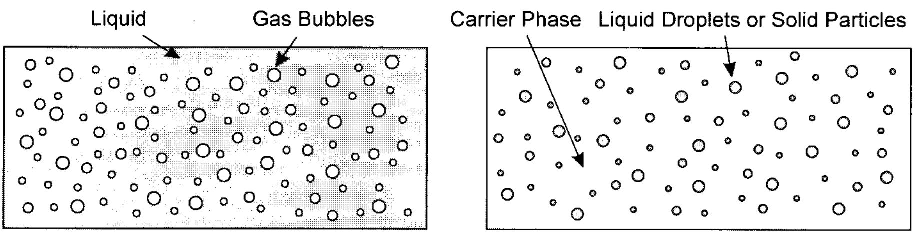

Fig. 1 Dispersed Two-Phase Flow – bubbly flow (left), particulate or droplet flow (right)

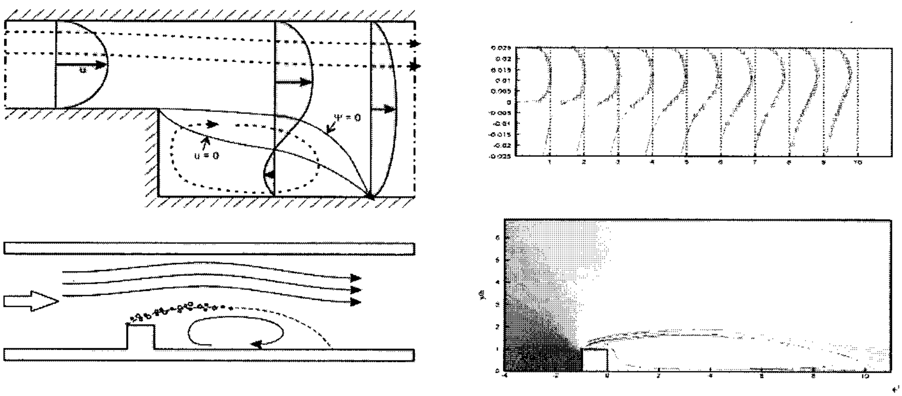

Fig. 2 Test Cases: Particulate flow (top), Bubbly flow (bottom)

REFERENCES

CHANSON, H.; TOOMBES, L. (2002): *Air-water flows down stepped chutes: turbulence and flow structure observations.* In: International Journal of Multiphase Flow, Vol. 28, pp. 1737 – 1761.

FARREL, K.J. (2003): *Eulerian/Lagrangian Analysis for the Prediction of Cavitation Inception.* In: Journal of Fluids Engineering, Vol. 125, pp. 46 – 52.

KUIPERS, J. A. M.; VAN SWAAJI, W.P.M. (1998): *Computational Fluid Dynamics applied to Chemical Engineering.* In: Advances in Chemical Engineering, Vol. 24, pp. 227 – 328.

SOMMERFELD, M. (2000): *Overview and Fundamentals Part I and II.* In: Theoretical and Experimental Modelling of Particulate Flow: April 3 – 7, 2000. /Ed. By J.-M. Buchlin. Rhode Saint Genèse: von Karman Institute for Fluid Dynamics (Lecture Series, 2000-06), pp. 1 – 63.