Mathematical modeling and numerical simulation of multiphase flows in metrology

The aim of the EMPIR project “Multiphase flow reference metrology” is to explain and reduce the uncertainty in multiphase flow metering in the oil and gas industries. Therefore, a typical multiphase flow measurement set-up consisting of a 16 meter long horizontal pipe followed by a relatively complex measurement unit is examined experimentally and numerically. Within the working group 8.41, a variety of industrially relevant configurations with different oil, water, and gas flow rates are simulated with the commercial CFD code ANSYS Fluent. Depending on the prescribed superficial velocities of the gas and liquid phases, different flow patterns are observed at the end of the inflow section, which have an influence on the accuracy of the Venturi meter. The CFD simulations allow a visualization of the different structures in all parts of the geometry, even in areas that can hardly be observed in experiments. Furthermore, the influence of different parameters (like the use of different fluids in the laboratories taking part in the experimental intercomparison of the project) on the pressure measurement in the Venturi tube has been investigated. An advantage of the simulation over the experiment is that it is possible to change only one parameter and keep the others constant. Thus, the influence of the different parameters can be investigated separately.

The CFD models have been validated by comparison of the results with data from the literature as well as with several test cases investigated in the experimental intercomparison of the project. Fig. 1 shows kerosene-nitrogen slug flow for one of the test cases as observed in the simulation with Fluent (on the left) and in the experiment at NEL (on the right). A comparison of the results shows a very good qualitative agreement. The structure of the slug is reproduced very well by the numerical simulations. Furthermore, also the time differences between the beginning, middle, and end of the slug match with the experimental observations. Both in the numerical simulation and in the experiment, one observes smaller waves behind most of the slugs. Further details can be found in [1].

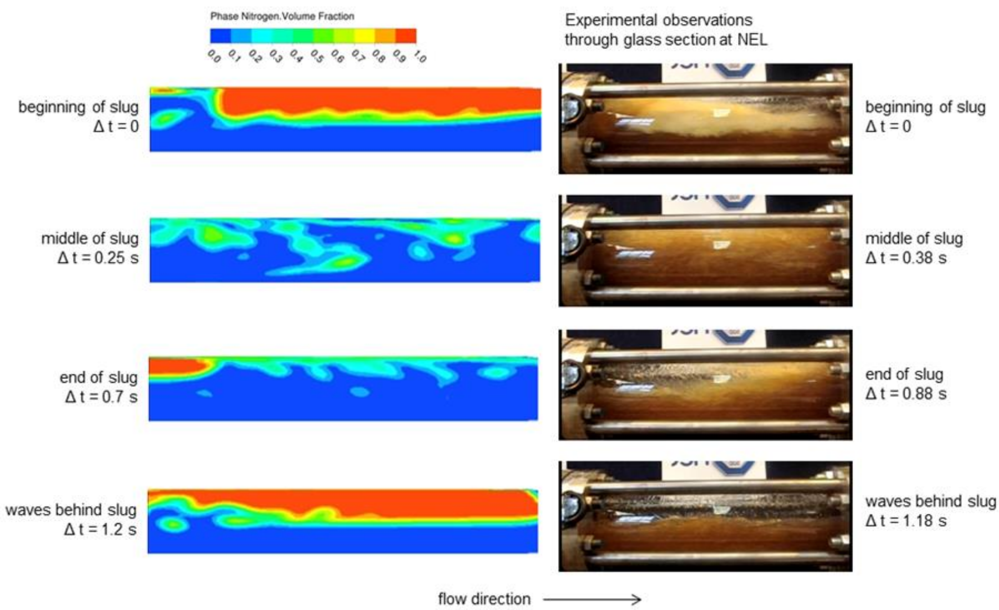

Figure 1: Comparison between simulation results obtained with Fluent (left) and experimental observations of the flow pattern through a glass viewing section at NEL (right).

In order to investigate the influence of several operational parameters on the pressure measurement, not only a horizontal pipe, but the whole transfer package, used in the experimental intercomparison of the project, was considered also in the CFD simulations. Fig. 2 shows the flow pattern for one test case simulated with Fluent. One observes the change from slug flow in the horizontal inflow section towards annular-like flow in the vertical Venturi tube. As expected, the blind-T leads to a mixture of the two phases.

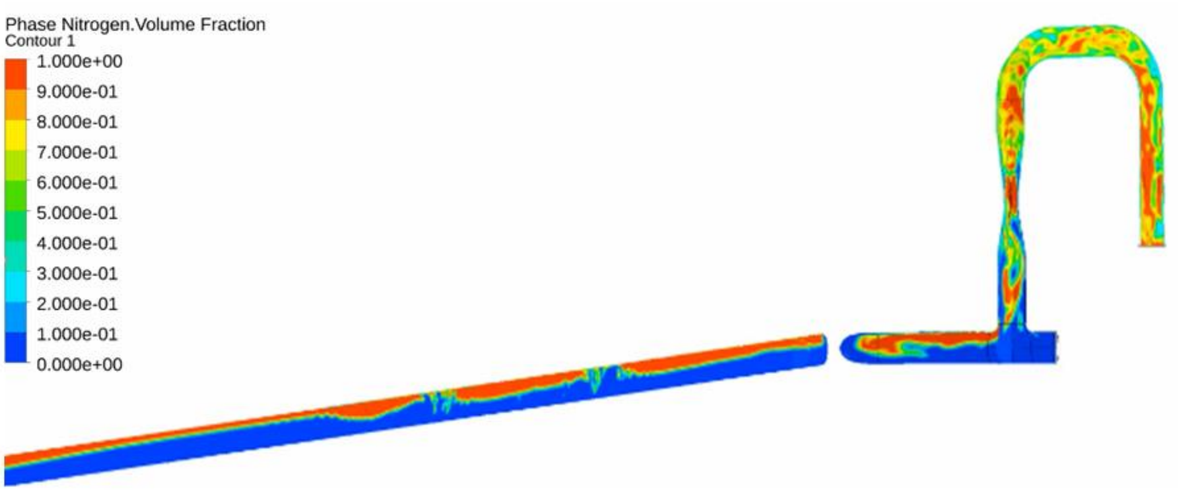

Figure 2: Simulation of one of the test cases: Development of slug flow in the horizontal inflow section (left) and mixture of the phases due to the blind-T (right).

For the comparison with experimental data, the pressure difference over the Venturi tube was calculated from the results of the CFD simulation. Fig. 3 shows the computed extremal and mean values of the pressure differences for several cases in comparison with experimental results.

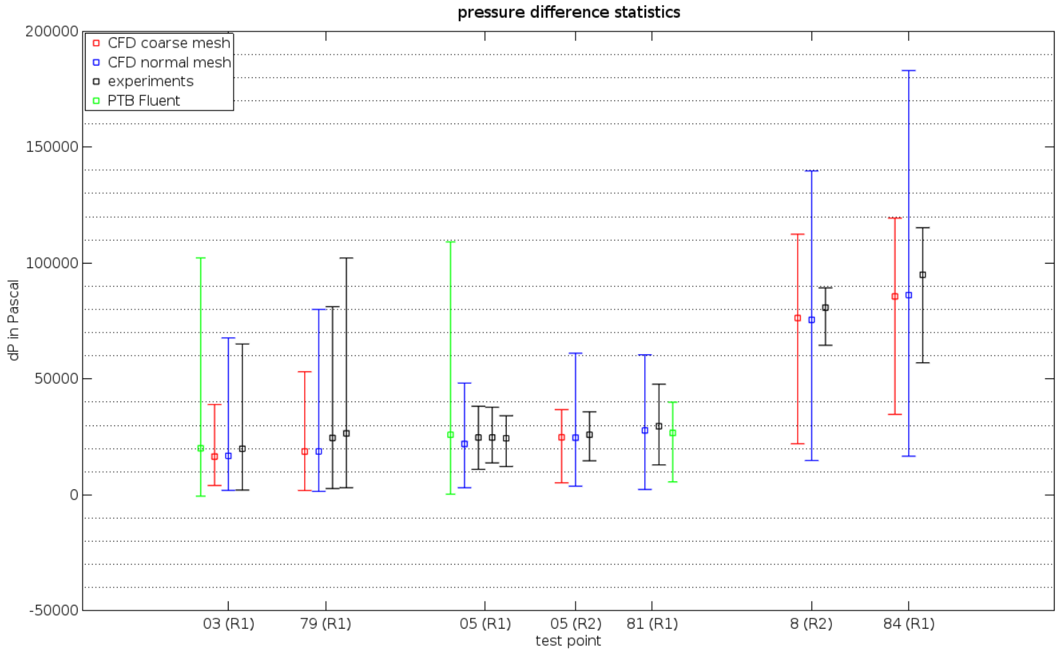

Figure 3: Comparison of extremal and mean values of the pressure difference (dP) by CFD predictions and by experimental observations for several test cases.

One observes good agreement of the mean values for both, the OpenFoam results obtained at the Czech Metrology Institute (CMI) (depicted in red and blue in Fig. 3) and the Fluent results obtained at PTB (shown in green).

The OpenFoam results underpredict the mean pressure difference slightly for all cases. The Fluent results, on the other hand, show a much wider spread between minimum and maximum pressure difference than observed in the experiment (see green lines for test point 03 and 05 in Fig. 3). The explanation is that much smaller time steps are used for the numerical simulations compared to the sampling rate of the Venturi meter. In order to resolve this difference, the simulation data have been re-sampled with the same frequency as the experimental data have been recorded. The corresponding results are shown in Fig. 3 for one of the test configurations (test point 81) leading to a much smaller range of pressure differences.

The influence of several parameters (like oil / gas density, oil / gas viscosity, surface tension, gas volume flow rate) has been investigated during the project, see also [2]. An advantage of the simulations is that it is possible to change only one of the parameters and keep the others constant, which can often hardly achieved in experiments. Thus, the influence of the different parameters can be investigated separately. As expected, the pressure difference in the Venturi tube increased with increasing liquid or gas density. On the other hand, a clear dependency on liquid or gas viscosity was not observed.

Literature

[1] A. Fiebach, E. Schmeyer, S. Knotek, and S. Schmelter. Numerical simulation of multiphase flow in a vertically mounted Venturi flow meter. In *Proceedings of the 17th International Flow Measurement Conference FLOMEKO 2016*, Sydney, Australia, September 26–29, 2016.

[2] S. Knotek, A. Fiebach, and S. Schmelter. Numerical simulation of multiphase flows in large horizontal pipes. In *Proceedings of the 17th International Flow Measurement Conference FLOMEKO 2016*, Sydney, Australia, September 26–29, 2016.