CFD SIMULATION AND EXPERIMENTAL VALIDATION OF FLUID FLOW IN LIQUID DISTRIBUTORS

Marc Heggemann[1], Sebastian Hirschberg[1], Lothar Spiegel[2], Christian Bachmann[2]
[1]Sulzer Innotec, Sulzer Markets & Technology AG, Fluid Technology, P.O. Box, CH-8401 Winterthur, Switzerland (E-mail: marc.heggemann@sulzer.com)
[2]Sulzer Chemtech AG, Column Technology Services, Sulzer Allee 48, CH-8404 Winterthur, Switzerland

KEYWORDS: CFD, VOF, distributor, packed column, orifice coefficient, validation

INTRODUCTION

The prediction of fluid flow through orifices of liquid distributors represents a challenging task. Numerous parameters like liquid height h, geometrical orifice area A and especially lateral velocity influence the volumetric outflow $\dot{V}$. The flow is not able to turn immediately at the sharp-edged corner of the orifice. The direction change of the horizontally approaching liquid may not be discontinuous. The result is a separation of flow within the orifice and a smaller hydraulic cross section relatively to the geometrical one. Therefore, the analytical outflow equation according to Bernoulli contains an orifice coefficient μ which takes the jet contraction into account [1]:

$$\dot{V} = \mu \cdot A \cdot \sqrt{2 \cdot g \cdot h} \tag{1}$$

g represents the acceleration of gravity. The detailed local intensity of cross flow is unknown in advance. As a result, CFD calculations are valuable due to the non-predictability of the individual fluid flow phenomena through specific orifices.

Furthermore, experimental data of the performance of liquid distributors are collected on test rigs using water as the liquid phase. Other liquids, especially liquids with higher viscosities, may have a different behaviour than water. On the one hand, experiments with other fluids are much more expensive due to the experimental setup. On the other hand, CFD simulations can be done easily. For this reason, it is interesting to have simulation capabilities in this matter, too.

This study investigates the capability of simulation methods to estimate the behaviour of liquid distributors using increasing geometrical complexity from a single orifice to a complete liquid distributor as operated in process industries. The uniformity of fluid flow characterizes the performance of liquid distributors. The local flow rate through the orifices is a function of the orifice coefficients (equation (1)).

SINGLE ORIFICE

As a first step, the flow through a single orifice was calculated. The CFD simulations are carried out in *FLOW-3D*® [2] due to its excellent free surface treatment which plays a

Table 1. Liquid properties and setup

Property	Value	Unit
Density ρ	1,000	kg/m^3
Viscosity μ	0.001	Pa·s
Surface tension σ (water – air)	0.070	N/m
Contact angle α (water – steel)*	80	°
Orifice diameter	2, 4, 8, 16	mm
Wall thickness	2	mm
Liquid height above orifices	60...400	mm

*Sulzer internal report

major role regarding the flow phenomena of liquid distributors. Water with its properties is used as liquid for the simulations (Table 1) and is compared to Sulzer internal experimental data for 2 manufacturing types of orifices: Lasered orifices (sharp-edged) and punched orifices (round-edged). Geometries used in simulations show typically sharp-edged contours. This corresponds to lasered orifices.

The numerical setup with regards to grid resolution, solver type, turbulence model and further boundary conditions was optimised. The grid around the orifice has to be comparably fine for correct numerical estimation. The standard k turbulence model proved to be sufficient. In case of smooth wall all over, the volumetric flow rate through the orifices would be more than 10% too high. The introduction of a ring around the orifices possessing surface roughness in the numerical setup represents the influence of the manufacturing process of the orifices (Figure 1). In consideration of this, the numerical results could be validated by experimental data. The degree of roughness depends on the liquid height and the orifice diameter.

The investigated orifice diameters d are between 2 and 16 mm which represent realistic values of orifices of liquid distributors in operation. The width of the surface roughness containing rings is constraint to 2 mm. Tests with adjusted ring widths proportional to the orifice diameters showed a merely small effect. For instance, the application of a surface roughness of 0.0005 m gives the best results using an orifice diameter of 4 mm at a liquid height of 200 mm. If the orifice diameter is 8 mm the surface roughness has to increase to 0.0025 m.

SINGLE CHANNEL

The investigation of a single channel containing 10 orifices characterizes the next degree of complexity (Figure 2). The orifice diameter is set to 4 mm, the wall thickness remains at 2 mm. The roughness is applied according to Chapter 2.

This geometry shows 2 types of feeding: The first has a central inlet on the top wall (symmetric case); the second is fed by an inlet at the side of the top wall (asymmetric case). Each inlet has a diameter of 12 mm. 6 cases without and including baffles are analysed; the 2 basic cases are discussed in detail in this chapter. The numerical

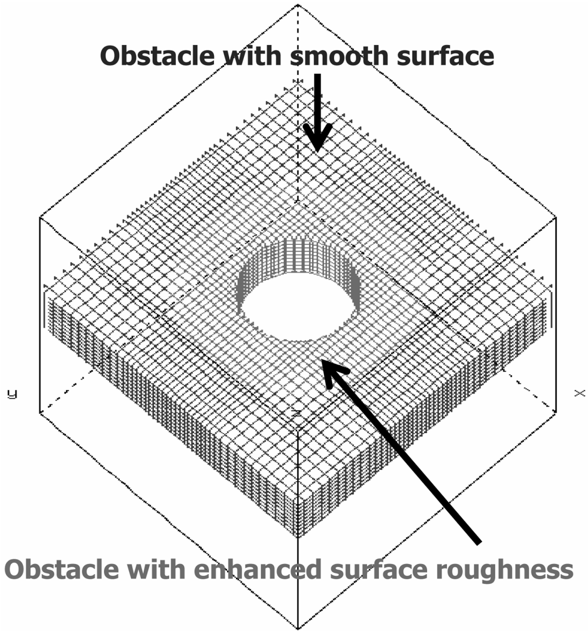

Figure 1. Ring around the orifice possessing surface roughness

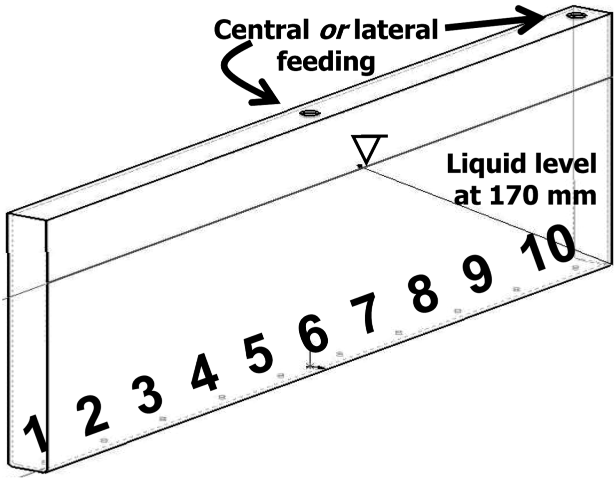

Figure 2. Single channel with central or lateral feeding (26 × 600 × 230 mm)

and experimental orifice coefficients are still in the focus of interest. A quasi-steady filling and outflow process is crucial to get a feedback of the approaching flow near the orifices. Thus, a real time flow of several seconds is required.

At first, an over-all consideration of in- and outflow in one simulation was tested. However the calculation times are huge: 1 hour CPU time on 4 DEC alpha processors (each 633 MHz) is needed per approximately 4 ms real time. This order of magnitude is not practicable for the analysis of the flow behaviour in the range of some seconds. Therefore, a 2-step approach was developed. The first simulation of this method calculates the quasi-steady state of fluid flow. The orifices are substituted by sinks which remove the liquid equally. The amount is estimated based on experimental results. A coarser grid can be used covering the more macroscopic flow phenomena. Consequently, the calculation time is acceptable. A quasi-steady state is reached at approximately 2 s real time which is assessed based on the lateral velocity component along the orifices. In the second simulation, the orifices are open again and a finer grid especially in the orifices areas is needed. This simulation covers the more microscopic fluid flow and is able to predict the individual orifice coefficients.

CENTRAL FEEDING

The volumetric flow rate and therefore the orifice coefficient are strongly dependent on the position of the orifices if no baffle avoids the direct feeding onto the orifices. Using a central feeding, the orifice coefficients are lowest for the centre orifices 5 and 6 due to the central inflow of the distributor and the resulting high lateral velocities close these orifices. In simulation, the orifice coefficients of the remaining 8 orifices 1 to 4 and 7 to 10 have a very similar value around 0.765 (Figure 3).

The fundamental behaviour of the simulated results (diamonds) could be validated by experimental data (circles). It has been found in the different test series that the outflow is sensitive to the exact location and orientation of the feeding pipe. In addition, air bubbles could not be totally avoided despite the optimised test rig setup. It has to be assumed that the remaining slight disagreement is particularly caused by the remaining bubbles in the vessel. The reasons are physically: On the one hand, the inlet water from the water supply has still some air bubbles. Furthermore, the free jet accelerates the adjacent air by drag forces. This leads to submerged air bubbles in the liquid bulk. These bubbles relocate the momentum of the free jet within the liquid. Consequently, the lateral velocities above the two central orifices are lower which lead to a higher outflow and higher orifice coefficients. The flow situation at the adjacent orifices is vice versa. On the other hand, the numerical model focus on the liquid phase using the VOF method, i.e. the gas phase is neglected (cells containing gas only are deactivated).

LATERAL FEEDING

The inflow region is moved from the centre to the side getting a different flow characteristic. The mesh is assumed the same except a further refinement at the new outer inlet region. The highest outflow and therefore orifice coefficient are at orifice 10 due to the

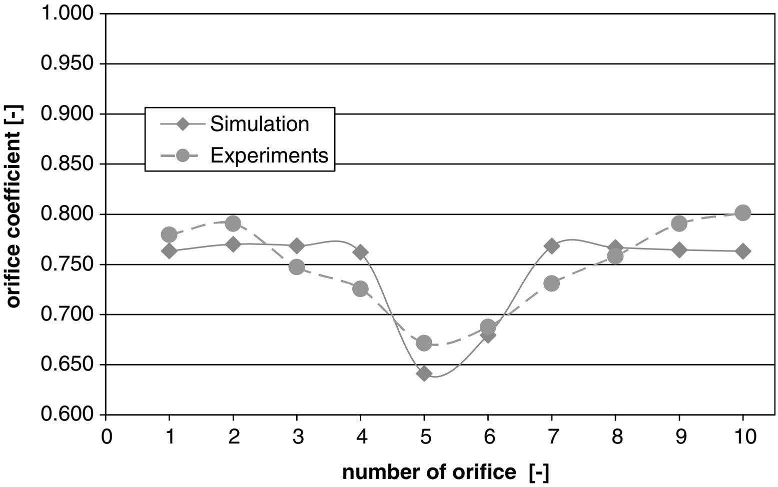

Figure 3. Central feeding with no baffles

direct approaching flow. The lowest orifice coefficient is right next to it at orifice 9 due to the high lateral velocity. The orifice coefficients of the remaining 8 orifices are increasing from orifice 8 down to 1. However, the values are in the same order of magnitude between 0.74 and 0.77 in simulation (Figure 4).

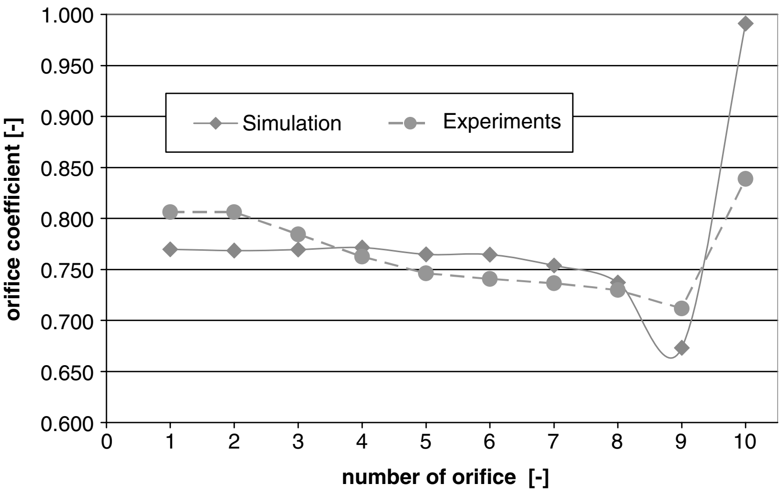

Figure 4. Lateral feeding with no baffles

Moreover, a sound validation of the simulation data (diamonds) could be performed in the tests (circles). In accordance with the experience described in Chapter 3.1, the differences in the numerical and experimental results can be explained by the slight bubbly flow in the test rig setup.

DISCUSSION

The two cases represent a real single channel distributor under special conditions, namely: feeding is straight directed at orifices which is avoided in general and baffles are used for this reason. The free jet is destroyed by devices like baffles or inlet calming boxes in commercial liquid distributors. Consequently, the fluid flow is already more homogeneous at the orifices and a slight bubbly flow has not that remarkable influence described above. Therefore, the 2 presented cases characterize exceptionally difficult simulation cases and the validation is already acceptable. *FLOW-3D*® is able to predict the individual orifice coefficients considering surface roughness introduced in Chapter 2. Accordingly, the investigation and validation of more complex liquid distributors at more realistic conditions will be performed.

CHANNEL DISTRIBUTOR INCL. PRE-DISTRIBUTOR

The crucial question: Is it possible to investigate orifice coefficients of complex liquid distributors by simulation in a reasonable calculation time range? The examination of a channel distributor including pre-distributor having 160 orifices represents the next degree of complexity (Figure 5).

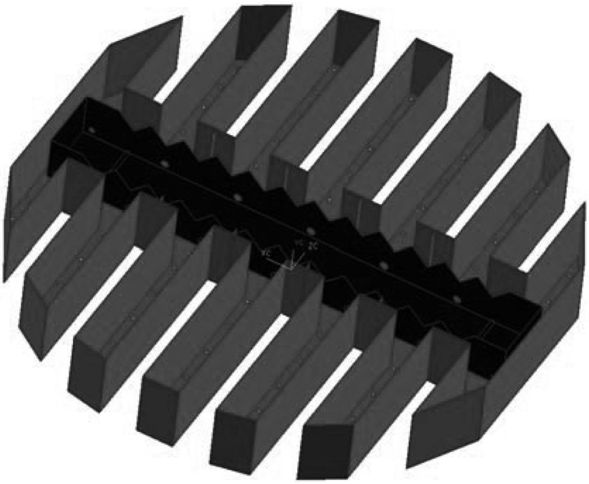

Figure 5. Channel distributor including pre-distributor

The orifice diameter is 8.2 mm, the wall thickness remains at 2 mm. The column diameter is 1,572 mm. The surface roughness is implemented in accordance with Chapter 2. This configuration is fed by 1 central inlet tube having an inner diameter of 150 mm. The most interesting operation conditions are the minimum load (59.9%) because of the low liquid height and the nominal load (100%). This channel distributor is in operation in a column since 2005. The corresponding simulations characterize a feasibility study.

EXPERIMENTAL SETUP

The channel distributor was tested on the distributor rig and all experiments were run using water as liquid phase according to the experimental tests of the single orifice (Chapter 2) and single channel (Chapter 3). Now, symmetrical aspects are taken into account due to the high orifices number of the channel distributor. The outflow was determined by area measurements at all 40 orifices of one quarter of the device which represents the entire distributor because of its geometrical symmetry. In addition, the symmetrical flow behaviour was assessed by random testing in the remaining three quarters of the channel distributor. Furthermore, the results are evaluated statistically.

NUMERICAL SETUP

All support plates as well as the pre-distributor including lateral orifices are considered and assembled by infinite thin baffles. As a result, the mesh size can be reduced. In addition, the inlet tube is neglected. Merely a mass source is located at the end of inlet pipe position as a further simplification step. As described in Chapter 3, the developed two-step approach is applied. At first, the 160 orifices are replaced by sinks and the quasi-steady state of fluid flow is calculated. The incoming liquid leaves the device equally distributed

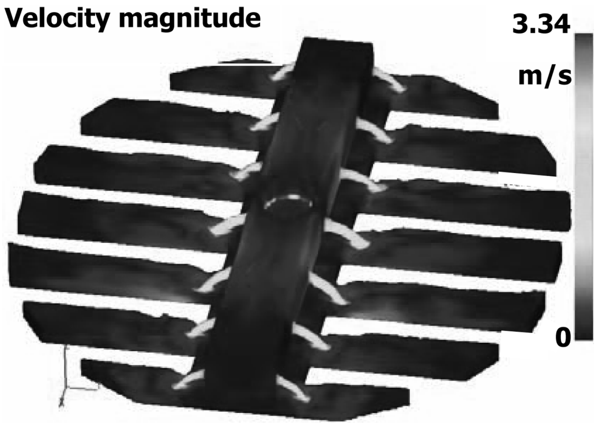

Figure 6. Fluid flow on a channel distributor including pre-distributor

through the 160 sinks. The liquid density, viscosity, surface tension and contact angle are chosen in accordance with the simulations in Chapters 2 and 3 (Table 1).

MINIMUM LOAD

The minimum load case represents the most critical setup due to the comparably low liquid height of merely 33 mm in the main distributor. The corresponding liquid flow rate is 18.75 m^3/h. The simulation is time-consuming due to the required time step. The fluid convection in axial direction is limiting stability which is caused by a dynamic free surface and the free jets between the pre- and main distributor. A snap shot of the fluid dynamics is presented (Figure 6).

The time when the quasi-steady flow is established has to be assessed quantitatively. Therefore, the lateral velocity component along the main flow direction close to the bottom above orifices of interest (Figure 7) is analysed.

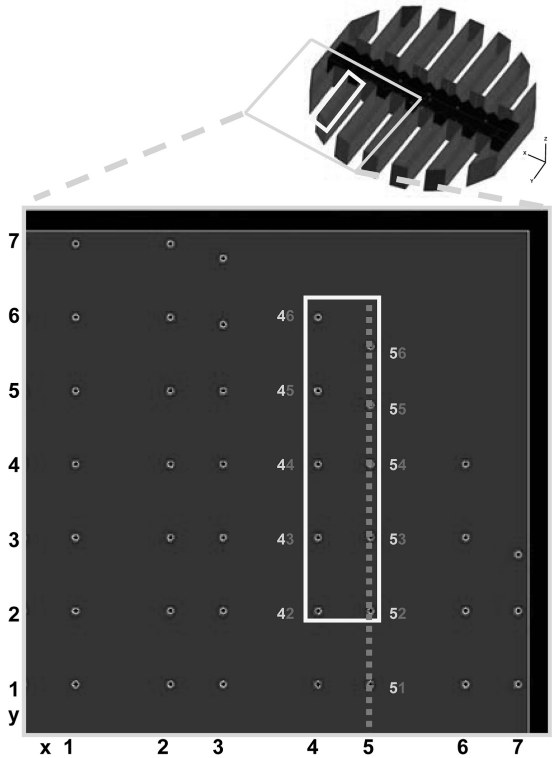

Figure 7. Quantitative assessment of quasi-steady state fluid flow along dotted line

A quasi-steady flow field based on this method is reached not later than at 3 s of real time. Slight fluctuation cannot be avoided. Several simulations using different meshes were carried out. The optimised mesh allows comparably quick simulations with good accuracy. The propagation of the lateral velocity component vs. time is shown (Figure 8). An optimisation of the mesh is in particular important for liquid distributor simulations. On the one hand, the numerical results may not be mesh dependent; on the other hand, the simulation time has to be reduced as much as possible with regard to the complex devices.

When a quasi-steady state is reached, the second step of the two-step approach can be carried out. The detailed outflow through the orifices of arbitrary parts of interest of the channel distributor is accessible by simulation considering the quasi-steady flow field. The sinks are replaced by orifices with same diameter (ø 8.2 mm) including 2 mm wide rings around. Surface roughness of 0.0025 m is added according to orifice diameter and liquid height (Chapter 2). As a first run, it is focused on the second arm due to the required high mesh resolution (small rectangle in Figure 7).

Similar outflow is reached after 50 ms starting at 2 s, 3 s and higher respectively. The outflow of the first mentioned time is presented to give an impression of the flow (Figure 9). The simulation results and experimental data are compared (Figure 10) and demonstrate a good agreement. The orifice coefficients are calculated using local liquid heights for the simulations and a mean liquid height of 34 mm for the experiments, respectively. In the experiments, the liquid height was measured manually. The accuracy can be estimated to $+/-$ 2 mm due to dynamic surface effects.

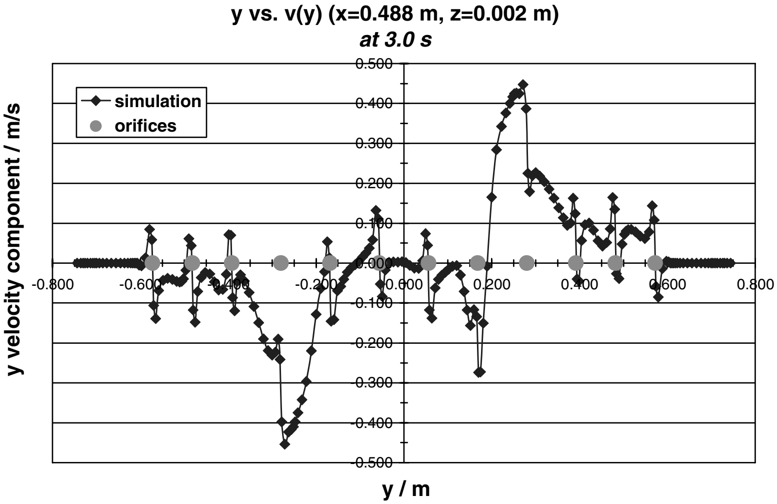

Figure 8. Quantitative assessment of lateral velocity acc. to Figure 7 using different grids

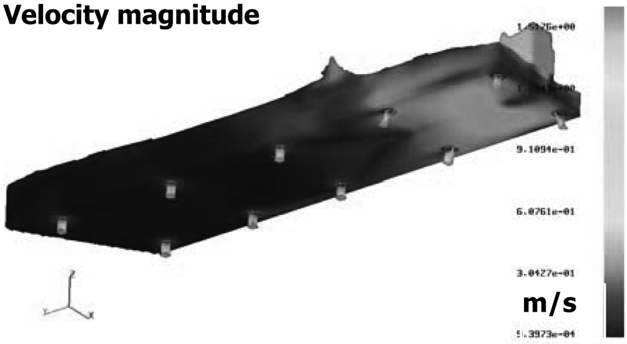

Figure 9. Fluid flow through the 10 orifices of the second arm after 2.05 s at minimum load

Experimentally, bubbly flow cannot be avoided totally. The pre-distributor contains a small but certain amount of bubbles which results in a slight bubbly jet flow from the pre- into the main distributor (Figure 11) which may influence the results to some extent.

NOMINAL LOAD

The nominal load case describes the mostly used configuration. The liquid height is tripled to 99 mm at 31.3 m^3/h liquid flow rate. The calculation of the quasi-steady flow field is in

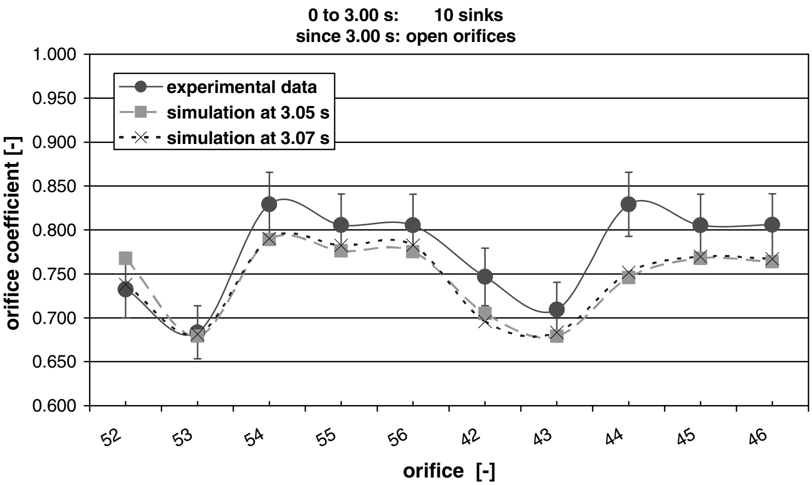

Figure 10. Simulated local orifice coefficients including experimental validation numbering refer to Figure 7

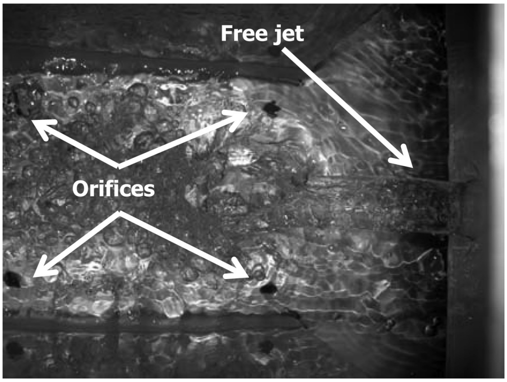

Figure 11. Free jet flow through lateral orifice of pre-distributor into the 2nd channel

the focus of step 1 of the two-step approach. Therefore, the mesh has to take into account the higher located free surfaces and the larger liquid heights in general. Consequently, the optimized mesh of the minimum load case is adapted in vertical z direction. The simulation time could be decreased distinctly in spite of a higher number of active cells for simulation. The reason for this is the approximately five times higher time step due to a

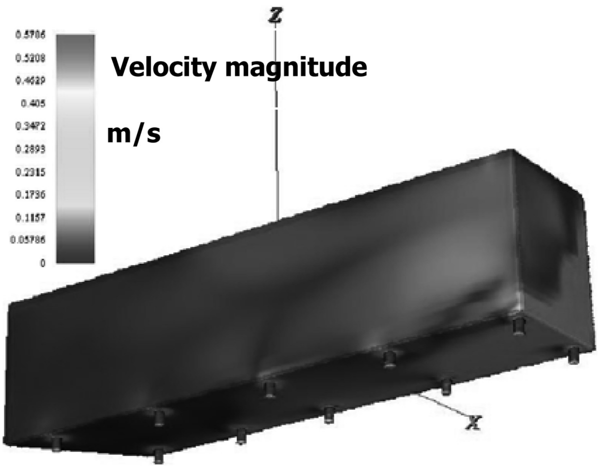

Figure 12. Fluid flow through the 10 orifices of the second arm after 2.05 s at nominal load

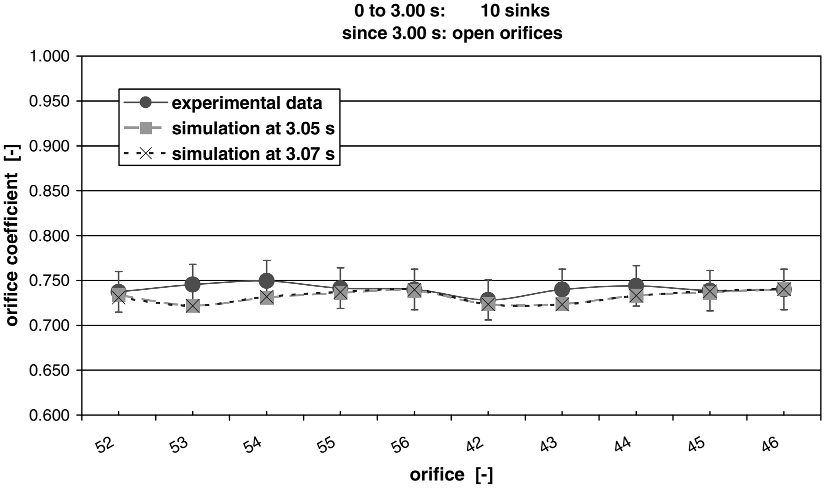

Figure 13. Simulated local orifice coefficients including experimental validation numbering refer to Figure 7

calmer free surface as well as missing free jets. The orifices of the pre-distributor are submerged at the nominal load conditions.

The outflow through the orifices has to be calculated in step 2. It is focused on the second arm of the channel distributor and the sinks are again replaced by orifices with 2 mm wide rings around in analogy to Chapter 0. Surface roughness is set to 0.0025 m, too (Chapter 2). The outflow after 50 ms in combination with a much larger liquid height is shown (Figure 12).

The simulation results and experimental data are compared (Figure 13) and demonstrate an excellent agreement.

CONCLUSIONS

The outflow rates through devices with increasing complexity have been analysed: Single orifice, single channel and channel distributor. Comprehensive experimental data have been generated. CFD simulations have been carried out. The methods have been fine-tuned. The introduction of a ring possessing surface roughness models the manufacturing process of the orifices. Therefore, a realistic estimation of the orifice coefficients is possible. Furthermore, a two-step approach has been developed and has been applied successfully to liquid distributor simulations. It allows the detailed calculation of local flow rates through orifices including orifice coefficients of extensive devices in a feasible expenditure of time.

The simulation results could have been validated: The single orifice and single channel data were investigated experimentally using constructions specifically built for this purpose. The channel distributor was tested on the distributor rig. The outflow was determined by area measurements using symmetry. The measuring tolerance in the testing can be quantified to 4.4% at minimum load and 3.0% at nominal load. The accuracy of the CFD simulations is in the same range.

The evaluation of the simulation method in liquid distributor development is scheduled as a next step. Fine-tuning of the method is needed.

REFERENCES

1. Jakubith, M., *Grundoperationen und chemische Reaktionstechnik*, Wiley-VCH, Weinheim, 1998.
2. *FLOW-3D*®, solver version 8.2.5, Flow Science, Inc., Santa Fe, NM, USA, 2005.