Conference Paper, Author's Postprint

Schulze, Lydia; Stamm, Jürgen; Thorenz, Carsten

A new two-phase flow model for the investigation of the effect of entrained air in navigation locks

Verfügbar unter/Available at: https://hdl.handle.net/20.500.11970/107588

Vorgeschlagene Zitierweise/Suggested citation:
Schulze, Lydia; Stamm, Jürgen; Thorenz, Carsten (2019): A new two-phase flow model for the investigation of the effect of entrained air in navigation locks. In: 38th IAHR World Congress : Water: Connecting the World; September 1-6, 2019, Panama City, Panama. Madrid: IAHR. S. 5763-5771.

Schulze, Lydia; Stamm, Jürgen; Thorenz, Carsten

A new two-phase flow model for the investigation of the effect of entrained air in navigation locks

Erstveröffentlichung in IAHR: E-proceedings of the 38th IAHR World Congress, September 1-6, 2019, Panama City, Panama. Madrid: 2019, S. 5763-5771.
Für eine korrekte Zitierbarkeit ist die Seitennummerierung der Originalveröffentlichung für jede Seite kenntlich gemacht.

S. 5763

A new two-phase flow model for the investigation of the effect of entrained air in navigation locks

Lydia Schulze [a]

Jürgen Stamm [b]

Carsten Thorenz [a]

[a] Federal Waterways Engineering and Research Institute, Karlsruhe, Germany
[b] Technische Universität Dresden, Dresden, Germany

Abstract

Air entrainment induced by plunging jets at ship locks is a widely observed phenomenon. Due to the lack of suitable investigation methods, the consequences of the air intermixing were unclear heretofore. In this study a new numerical approach for the modelling of multiphase flows with various interface scales is used to investigate the effects of air entrainment on the ship forces during the filling of a lock. The results show that the air entrainment has a significant influence on the flow regime in the lock chamber and can produce critical ship forces.

Keywords: air entrainment, ship lock, filling system, numerical simulation, OpenFOAM

1 Air entrainment at navigation locks

Many navigation locks in Germany with lifting heights lower than 10 m are filled through valves in the upstream gates. For some of them, a free falling jet evolves downstream of the valves which plunges into the chamber water body and entrains a significant amount of air. Due to buoyancy the air-water mixture rises to the water surface and accumulates in front of the ship. As a result, the water level in the chamber inclines and thus produces longitudinal forces on a ship in the lock chamber. The resulting forces can be larger than in situations without entrained air.

To enable the investigation of the effect of air transport processes on the ship forces, a new numerical approach was developed and implemented into the open source CFD-library OpenFOAM®. This study aims to explain the background of the developed approach and shows an example application. In particular, the effects of air transport and detrainment processes within an existing lock are investigated, where strong air entrainment was observed during the commissioning of the prototype. The numerical model examines the sensitivity of the ship forces to different air contents and various inflow rates.

Air entrainment during the filling process is a phenomenon that can be observed at navigation locks with various filling systems. Due to large head differences that have to be surmounted by the filling water, large flow velocities and strong air water intermixing can occur during the filling of some navigation locks. Through the large flow velocities the entrained bubbles cannot detrain at the inception point and thus, are transported into the filling system. Uncontrolled detrainment of the bubbles within the lock chamber can significantly influence the flow regime and the resulting forces on the ship. For the design of new locks and the optimization of existing structures it is essential to investigate the influence of the air entrainment on the system.

2 Design criterions and investigation methods for lock filling systems

To guarantee a safe filling process, the ship forces have to be limited when designing ship lock filling systems. The acting forces result mainly from water level slopes or surge waves evolving within the chamber during the filling. Too large forces can lead to breaking mooring lines, which represents a severe danger to the lives of the staff mooring the ship. In addition to optimizing the system for the ship forces, a good compromise between a short filling time and low building and maintenance costs has to be found when designing the filling system of a navigation lock.

Schulze, Stamm, Thorenz: A new two-phase flow model for the investigation of the effect of entrained air in navigation locks.
In: IAHR: E-proceedings of the 38th IAHR World Congress, September 1-6, 2019, Panama City, Panama. Madrid: 2019, S. 5763-5771.

S. 5764

Depending on the investigated question, various methods are used to optimize the hydraulic system: With state-of-the-art 3D-simulations the flow characteristics can be satisfactorily reproduced to answer many design questions regarding shapes of the filling system. For the evaluation of the ship forces physical scale models, analytical estimations or 1D-models (e.g. Belzner et al., 2018) are usually used, as the inclusion of a moving ship body within the narrow lock chamber in a 3D-simulation is very complex and currently still subject of research and development (Thorenz et al., 2017).

Due to the lack of suitable methods, the investigation of the effect of air entrainment in navigation locks was mostly ignored in past designs or the system was designed such that uncontrolled air entrainment was avoided, as recommended by Partenscky (1986). In Froude scaled models, which are often used for the optimisation of the filling times, scale effects regarding the entrained air cannot be avoided. But also the widely applied numerical investigation approaches based on the Volume-of-Fluid (VoF) method are not able to capture the effects of the air entrainment and transport processes physically correct with feasible computational resources. A satisfying examination of the effect of air entrainment is therefore not possible with these methods. Though, entrained air bubbles within a filling system of a lock can significantly change the flow regime and thereby increase the evolving forces on the ship.

3 A new approach for investigating the effect of air entrainment at ship locks

To enable the investigation of the filling process, all relevant physical processes have to be modelled in the numerical approach. Thus, a suitable model has to be able to capture multiple two-phase phenomena that occur during the filling. Especially the large range of interface scales presents a specific challenge: on the one hand the largely stretched water surface within the lock chamber has to be carefully represented, where small errors can have significant effect on the ship forces. On the other hand the small air bubbles, which are entrained through the plunging filling water, have to be accounted for in the numerical simulation.

For solving multiphase flow problems various models exist, which can be categorized in three groups: Euler-Euler approach models, Euler-Lagrange models and Free-Surface flow models. The first model group is designed for modelling dispersed flows. Both phases are treated as continuum with separated velocity and pressure fields for each phase. The interaction between the phases is modelled with the means of closure terms. In practice, the reliability of the model is dependent on the formulation of the model closures. Thus, suitable definitions for the interfacial momentum transfer as well as the turbulence closure have to be found for each flow regime. As there is no special treatment for large stretched interfaces, numerical diffusion smears those. The Euler-Lagrange models are made for investigating particle flow. Here, the one phase is represented as continuum and the second phase consists of dispersed Lagrange particles. A transition from continuous to dispersed or vice versa cannot be modelled. However, the model can be coupled with Free-Surface models. These Free-Surface models are designed for modelling two-phase flows with interfaces that are much larger than the edge length of the cells. The approach is based on the assumption that both phases share one velocity and one pressure field. The phase distribution is calculated with an advection equation.

In this study, the open source library OpenFOAM® was used. The framework provides a large variety of linear equation solvers, turbulence models and discretization schemes which can be reused for individual implementations. For the modelling of multiphase flows, Euler-Euler approach solvers like *twoPhaseEulerFoam* as described in Rusche, (2002), Euler-Lagrange models like *icoLagrangianFoam or MPPICFoam* and Free-Surface flow models like *interFoam* are available. In the free surface flow model *interFoam* a special algebraic Volume-of-Fluid (VoF) scheme, called MULES (described in Damian, 2013), is applied for the solution of the advection equation, which counteracts unavoidable numerical diffusion. Through the introduction of an artificial velocity normal to the gradient of the phase fraction distribution, the scheme ensures the relatively sharp reproduction of the interface between the phases. This artificial velocity produces unphysical behavior when unresolved dispersed structures are present. Additionally, the calculated mixture velocity and mixture pressure field in cells with unresolved dispersed structures is neither physical. As a result, the physical behavior of unresolved bubbles or droplets cannot be modelled appropriately. However, this is essential when investigating flows with free-surface and relevant disperse intermixing of the phases.

To overcome this limitation of the existing free-surface approach, Schulze (2018) developed and implemented a new numerical approach OpenFOAM® for the investigation of the effect of entrained air on the filling. The new approach is able to capture both, the large scale air-water interface in the chamber and the effect of the rising bubbles on the flow below the surface. As the investigation focussed on the air transport and

Schulze, Stamm, Thorenz: A new two-phase flow model for the investigation of the effect of entrained air in navigation locks.
In: IAHR: E-proceedings of the 38th IAHR World Congress, September 1-6, 2019, Panama City, Panama. Madrid: 2019, S. 5763-5771.

S. 5765

detrainment processes, the current implementation does not yet account for the physical capturing of small water droplets in the air region.

For capturing the large scale air-water interface the new solver extends the Volume-of-Fluid (VoF) method. The implementation is based on the incompressible VoF solver interFoam. As entrained air bubbles cannot be resolved with reasonable mesh sizes, the approach is extended to a mixture approach so that the momentum transfer between the phases is physically captured. The implementation is optimized for large hydraulic engineering applications including complex geometries and highly turbulent flows. Compressibility effects of the bubbles are neglected in the implementation.

The solver consists of the following three equations. The first equation describes the convective transport of the primary phase, the second equation defines mass conservation for the incompressible mixture and the third equation describes the momentum conservation for the mixture.

Mass conservation equation for the primary phase

$$\frac{\partial \alpha_1}{\partial t} + \nabla \cdot (\alpha_1 \mathbf{j}) + \nabla \cdot [\alpha_1 (1 - \alpha_1) \mathbf{U}_\mathrm{r}] = 0 \qquad (1)$$

where,

α_1 is the volume fraction of the first phase, α_2 is the volume fraction of the second phase.

$\mathbf{j}$ defines the velocity of the centre of mass (cf. equation 5) and

$\mathbf{U}_\mathrm{r}$ represents the relative velocity between the phases (cf. equation 4).

Mass conservation equation for the mixture

$$\nabla \cdot \mathbf{j} = 0 \qquad (2)$$

Momentum conservation equation for the mixture

$$\frac{\partial \rho_m \mathbf{U}_\mathrm{m}}{\partial t} + \nabla \cdot (\rho_m \mathbf{U}_\mathrm{m} \mathbf{U}_\mathrm{m})$$

$$= -\nabla_{\mathrm{p_{rgh}}} + \nabla \cdot [\mu_m (\nabla \mathbf{U}_\mathrm{m} + \nabla \mathbf{U}_\mathrm{m}^\mathrm{T})] + \mathbf{g}\rho_m \cdot \mathrm{x} + \nabla \cdot \left[\alpha_1 (1 - \alpha_1) \frac{\rho_1 \rho_2}{\rho_m} \mathbf{U}_\mathrm{r} \mathbf{U}_\mathrm{r}\right] \quad (3)$$

where,

$\rho_m = \alpha_1 \rho_1 + \alpha_2 \rho_2$ represents the density of the mixture of the phases,

$\mu_m = \mu_1 \rho_1 + \mu_2 \rho_2$ is the mixture viscosity,

g stands for the gravity vector,

x represents the spatial position vector.

$\mathrm{p_{rgh}}$ is the pressure, modified as follows:

$$-\nabla \mathrm{p} = -\nabla \mathrm{p_{rgh}} - \mathrm{g} \cdot \mathrm{x} \nabla \rho_m - \rho_m \mathbf{g} \quad (4)$$

The set of equations contains three different velocities: the velocity of the centre of mass **j** (cf. equation 5), the velocity of the centre of the volume $\mathbf{U}_m$ and the relative velocity between the phases $\mathbf{U}_r$ (cf. equation 4). A correlation between the velocity of the centre of volume and the velocity of the centre of mass reduces the number of unknown variables and makes the system solvable:

$$\mathbf{j} = \mathbf{U}_m - \alpha_1 (1 - \alpha_1) \frac{\rho_1 - \rho_2}{\rho_m} \mathbf{U}_r \quad (5)$$

Through the factor $\alpha_1(1 - \alpha_1)$ the relative velocity only acts in cells, where both phases are present. The relative velocity for bubbly flow $\mathbf{U}_r$ can be approximated by the magnitude of the terminal rising velocity $\mathrm{U_t}$ of the bubbles multiplied with a unit vector pointing in opposite direction to the gravitation vector.

$$\mathbf{U}_r = \mathrm{U}_t \cdot \left[\frac{-\mathbf{g}}{|\mathbf{g}|}\right] \quad (6)$$

Schulze, Stamm, Thorenz: A new two-phase flow model for the investigation of the effect of entrained air in navigation locks.
In: IAHR: E-proceedings of the 38th IAHR World Congress, September 1-6, 2019, Panama City, Panama. Madrid: 2019, S. 5763-5771.

S. 5766

For bubbly flow the relative velocity between the phases can be interpreted as the bubble rising velocity. In the implementation, there is a choice between four approaches for the calculation of the terminal rising velocity dependent on the bubble diameter. The example below applied the approach by Zheng and Yapa (2000). Based on the knowledge that very small bubbles (bubble diameter $d_b < 0.001\ m$) form a spherical shape, with growing diameter ($0.001 < d_b < 0.02\ m$) the shape gets ellipsoidal and large bubbles ($d_b > 0.02\ m$) form a cap-shape, their approach consists of three empirically derived formulae for the three shape regimes to calculate the bubble rising velocity.

For simplicity the bubble diameter is assumed to be constant throughout the domain and the bubbles are assumed to reach the terminal rising velocity within a very short time so that acceleration can be neglected in the simulation.

The equation system is of second order, as it contains the second derivative of the mixture velocity $\mathbf{U}_m$. The momentum equation is non-linear due to the non-linearity of the convection term. With the given formulation a dependency between the mass and the momentum conservation equation is not directly visible, although it exists. A sophisticated solution algorithm allows solving the system. Details of the algorithm can be found in Schulze (2018).

For the discretisation of the set of equations the finite volume method formulated for unstructured meshes (as described e.g. in Jasak, 1996) is used. The OpenFOAM® library provides a large variety of discretisation schemes, which can be used for the discretisation of the terms of the equations. To counteract numerical diffusion and avoid under and overshoots when solving the scalar transport equation (equation 1), which calculates the water air distribution within the simulation domain higher order schemes in combination with the flux-corrected transport algorithm MULES (details described in Damian, 2013) are applied. For the solution of the mass and momentum equation a pressure-velocity coupling algorithm is applied. The so-called PIMPLE algorithm is a combination of the widely used PISO algorithm (Pressure-Implicit with Splitting of Operators by Issa, 1985) and the SIMPLE algorithm (Semi-Implicit Method for Pressure Linked Equations by Patankar, Spalding, 1972) algorithm, applicable for transient simulations and Courant numbers larger than unity. Subsequent to pressure-velocity coupling the equations of the turbulence model are solved, which model the effect of the velocity fluctuations in the flow without resolving all turbulent scales. The integration of the effect of the turbulence is achieved via the effective viscosity in the momentum equation (equation 3), which is calculated as sum of the molecular and the turbulent viscosity calculated in the turbulence model. The common VoF models are not designed for two-phase flows mixed by turbulence, therefore the interaction between the bubbles and the velocity fluctuations in the flow cannot be accounted for. To counteract this, the developed model was extended with an additional diffusion term in the transport equation for the gaseous phase.

$$\frac{\partial \alpha_2}{\partial t} + \nabla \cdot (\alpha_2 \mathbf{j}) + \nabla \cdot [\alpha_2 (1 - \alpha_2) \mathbf{U}_r] = b[\nabla \cdot (D_t \nabla \alpha_2)] \quad (7)$$

To create a connection between the diffusion term and the turbulence of the flow the dimensionless turbulent Schmidt number Sc_t is used, which defines the ratio of the turbulent diffusion ν_t and the turbulent diffusivity D_t.

$$Sc_t = \frac{\nu_t}{D_t} \quad (8)$$

For the simulations, the turbulent Schmidt number was set to unity, based on the assumption that the diffusion coefficient for the primary phase transport equals the turbulent diffusion. The user defined value *b* allows controlling the influence of the diffusion term. The turbulence extension of the phase equation can be used with RANS turbulence models of the OpenFOAM® library that are capable for incompressible flows. In future developments we aim to implement the impact of the density difference on the turbulence.

At the end of the solution routine of one time step variables are updated. Relying on the OpenFOAM® library structure the parallelization of the solver is inherent. This allows the simulation of large simulation domains as e.g. a complete lock chamber within reasonable time.

Schulze, Stamm, Thorenz: A new two-phase flow model for the investigation of the effect of entrained air in navigation locks.
In: IAHR: E-proceedings of the 38th IAHR World Congress, September 1-6, 2019, Panama City, Panama. Madrid: 2019, S. 5763-5771.

S. 5767

4 Example application: Lock Bolzum

4.1 The modelled lock

The following example shows the analysis of the effect of air entrainment on the ship forces in a typical through-the gate filling system with the new model approach. The simulated structure represents the existing lock of Bolzum, which connects the Mittelland Canal with the side canal of Hildesheim. With the lock a height difference of approximately 8 m can be surmounted. The chamber has a width of 12.5 m and a usable length of 139 m.

Figure 1 shows a sketch of the front part of the lock chamber with the upstream gate, the energy dissipation zone consisting of baffle blocks and a grid wall. For the filling, the upstream segment gate is slightly turned to release the upstream water into the lock chamber.

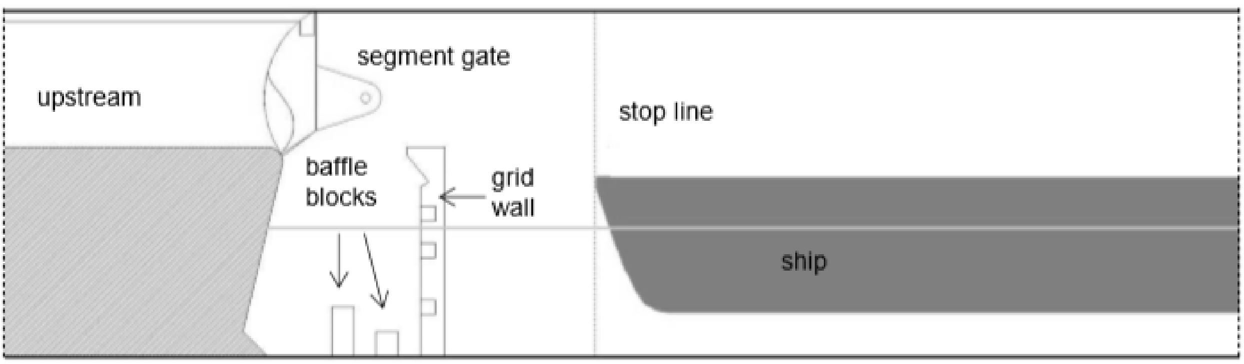


Figure 1: Sketch of the upstream part of the lock (adapted from Schulze, 2018).

In the beginning of the filling process a free falling jet evolves, which plunges into the energy dissipation zone in the front part of the lock chamber. Through the plunging a significant amount of air is entrained into the chamber water. Observations at the prototype showed that a large amount of the entrained air is transported into the region of the chamber, where the ships are supposed to moor during the lockage. Furthermore, high surface velocities were observed during the filling of the prototype lock.

4.2 The model

For the simulations a three dimensional model of the lock was used. To avoid the necessity of modelling a floating ship body, the vessel geometry was modelled as rigid body situated at the foremost position which is allowed during the lockage in the prototype lock. To create a quasi-steady state, the water level is kept constant by inserting vertical inflow into the system at the upstream side of the wall and releasing the same amount of water through the bottom outlet. The error that is introduced by this artificial velocity at the bottom (<0.02 m/s) is negligible in comparison to the main velocities in the model. The plunging jet was replaced by an inflow boundary patch, as the air entrainment phenomena in the falling jet cannot yet be reproduced with the model. At the inflow boundary, four different air contents (0 %, 10 %, 20 % and 30 %) and different flow rates (10 m^3/s, 20 m^3/s and 30 m^3/s) were prescribed.

Schulze, Stamm, Thorenz: A new two-phase flow model for the investigation of the effect of entrained air in navigation locks.
In: IAHR: E-proceedings of the 38th IAHR World Congress, September 1-6, 2019, Panama City, Panama. Madrid: 2019, S. 5763-5771.

S. 5768

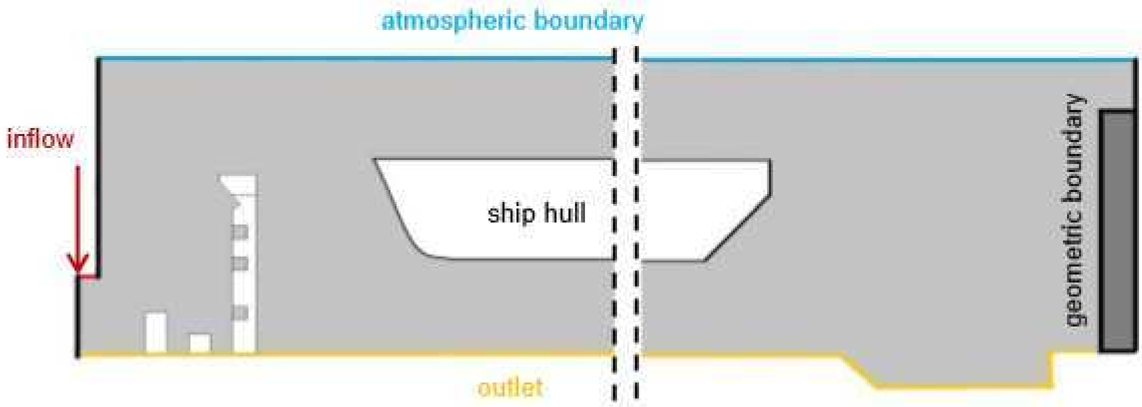


Figure 2: Boundary conditions for the numerical model of the lock (adapted from Schulze, 2018).

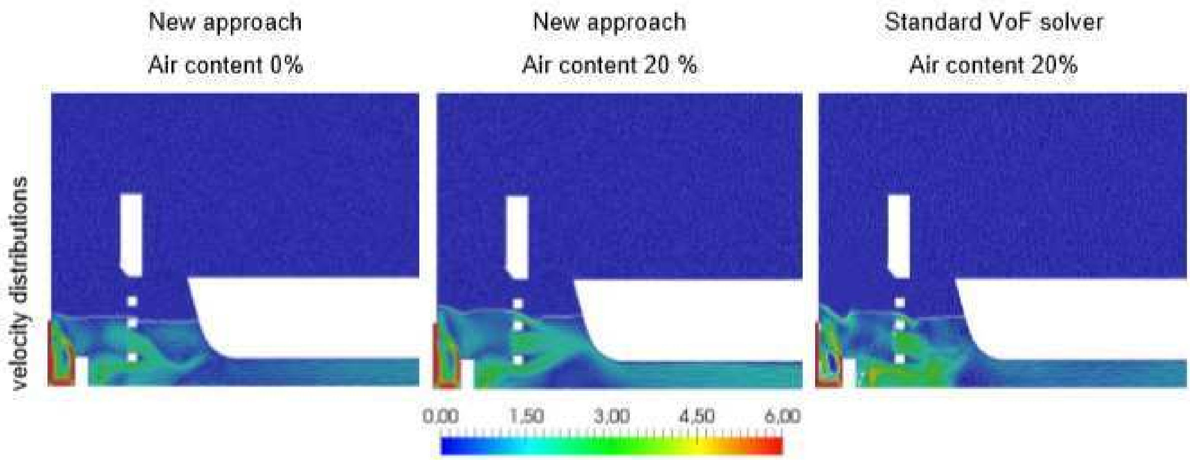
All geometry parts are modelled as walls, the top boundary is defined with an atmospheric boundary condition. The model is constructed as half model with symmetry plane along the longitudinal axis of the lock chamber. For clarification a sketch of the boundary conditions can be found in Figure 2.

The mesh used consisted of approximately two million cells, with a base edge length of 0.5 m. A finer mesh resolution with edge lengths of 0.25 m was used in the water filled region with additional refinements up to edge lengths of 0.0625 m in proximity of to the ship and the geometry walls. To allow an accurate definition of the inlet width very fine discretisation was necessary in the region around the inlet boundary. In a pre-study the mesh resolution was compared against two finer and one coarser mesh. The deviations in the resulting flow patterns in the velocity field, the pressure distribution as well as the air content transportation and degassing, between the standard case and the finer mesh cases were very small. Therefore, the above described resolution was chosen as good compromise between accuracy and computational effort.

To account for the turbulence, the k-omega-SST RANS (Menter, 1993) model was applied. After a total simulation time of approximately 800 s, changes in the flow field were small, so that the results can be considered as quasi-steady. Thus, this state is considered as steady state result for the visual analysis. For the quantitative analysis of the ship forces the simulations were performed for 1500 s and the values were averaged over the last 500 s.

4.3 The results

Figure 3a and Figure 3b present selected results of the new solver with two different air contents at the inlet. Figure 3c shows an exemplary result of the simulation with the standard OpenFOAM® VoF solver interFoam with 20 % air content at the inlet.



Schulze, Stamm, Thorenz: A new two-phase flow model for the investigation of the effect of entrained air in navigation locks.
In: IAHR: E-proceedings of the 38th IAHR World Congress, September 1-6, 2019, Panama City, Panama. Madrid: 2019, S. 5763-5771.

S. 5769

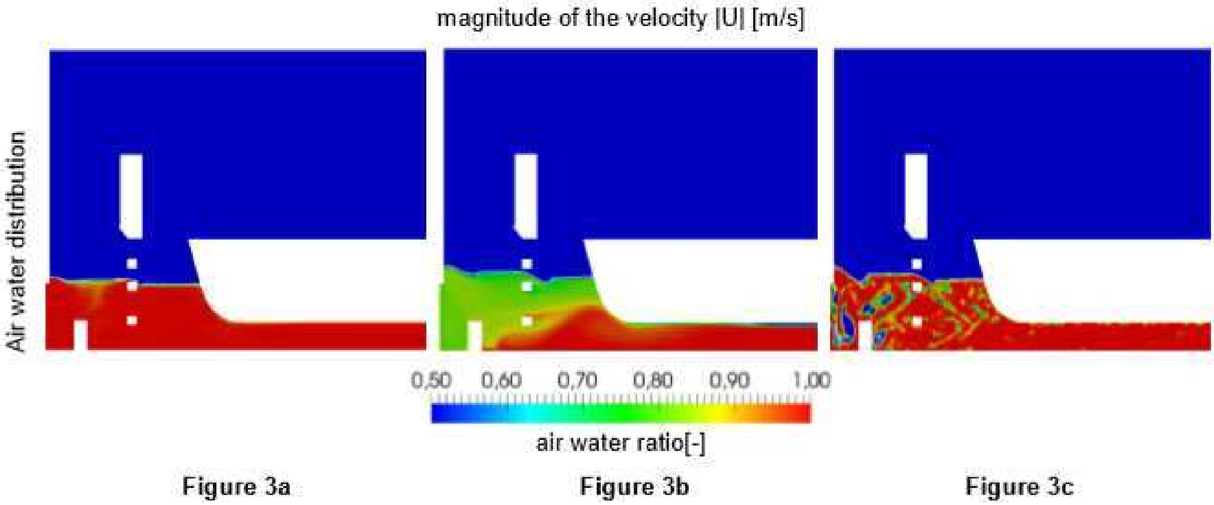


Figure 3: Visualization of selected results, showing the velocity field and the air-water distribution on a vertical plane.

When no air is entrained at the inflow boundary (Figure 3a), the energy is successfully dissipated through the formation of eddies in the energy dissipation zone. The flow behind the grid wall is homogeneously distributed over the height and the velocities at the ship hull are small. As desired the results with the standard VoF solver show almost no difference for the simulation without air entrainment. By contrast, the results with air entrainment at the inlet show very different flow patterns with the different solvers. In the results of the new approach (Figure 3b) the front part of the lock is filled with an air water mixture. Within the mixture region phases do not visibly separate, but the air water mixture, representing the unresolved bubbles, is continuously transported on a diagonal track towards the water surface, where the bubbles are detrained. The diagonal transport results from the vertical rising velocity of the bubbles and the horizontal velocity of the flow.

In contrary, the results of the VoF simulations (Figure 3c) show a fast separation of the phases in the energy dissipation zone. Large air-water mixture chunks appear which bear a certain similarity to super-large bubbles. The separation of the phases in the VoF approach is based on artificial compression terms, which are designed to counteract numerical diffusion at largely stretched air-water interfaces. In the case of unresolved bubbles, the resulting compression is not physically based.

The results of the simulation with the new solver show that the entrained air has a significant impact on the flow field in the lock chamber. When air is entrained at the inflow boundary, the flow behind the grid wall is deflected towards the water surface and the velocities get larger in the upper part of the wall and in proximity of the ship hull. Additionally, the water level rises in the air-water intermixing zone. With growing air contents, the jet in the upper part of the grid wall and the velocities at the ship hull get stronger. With increasing inflow rates, the effects intensify. Comparing simulations with the same air contents but different flow rates, stronger horizontal velocity components towards the ship are visible. As a result the bubbles are transported further into the chamber. The growing velocities towards the ship hull as well as the increasing water levels in the front part of the lock chamber result in increasing ship forces pointing in downstream direction.

Schulze, Stamm, Thorenz: A new two-phase flow model for the investigation of the effect of entrained air in navigation locks.
In: IAHR: E-proceedings of the 38th IAHR World Congress, September 1-6, 2019, Panama City, Panama. Madrid: 2019, S. 5763-5771.

S. 5770

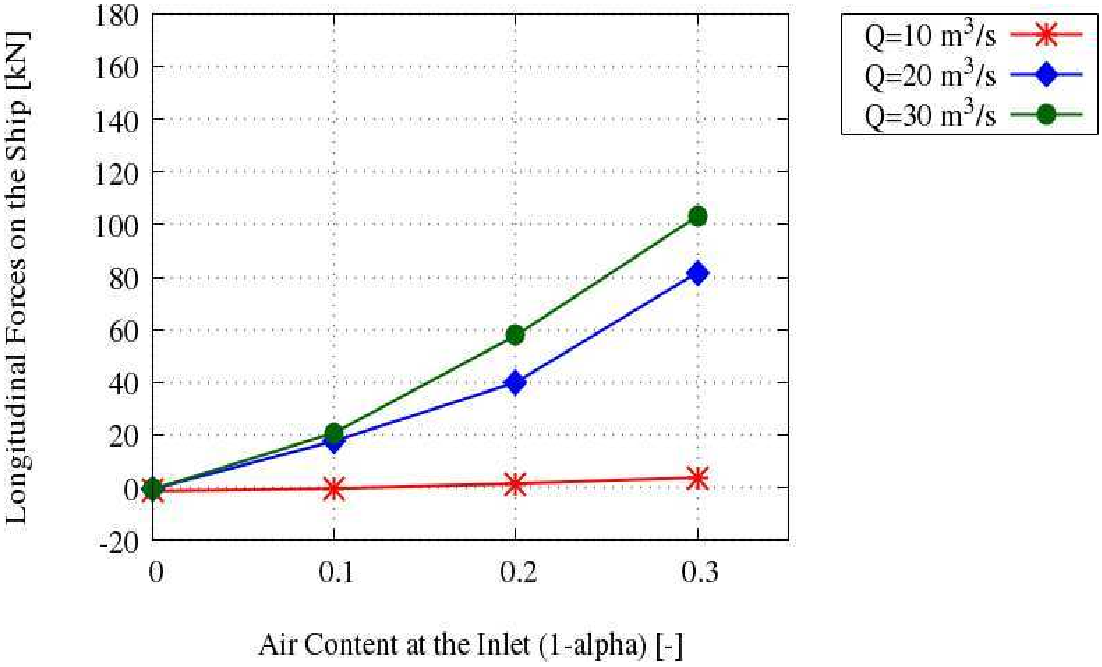


Figure 4: Forces on the ship dependent on the inflow rate and the inflow air content (Schulze, 2018).

Figure 4 depicts the evaluation of the ship forces for the tested inflow variants. The analysis of the quantitative results of the ship forces show that without air entrainment, the forces are very small. With growing air contents and growing inflow rates, the forces increase. According to the German regulations for locks (as described in Partenscky, 1986), the maximum forces on the ship should not be larger than 23 kN for the chosen setup. Comparing the resulting forces with these regulations it can be concluded, that for an inflow rate of 10 m / s air entrainment of up to 30 % does not produce critical additional forces. For larger inflow rates, the air content entrained through the plunging jet is the decisive factor.

As it is a hard task to measure resulting ship forces in prototype scale (due to large forces, required equipment and safety concerns), the model was not validated in terms of ship forces. The model approach without dispersed air has been validated for ship forces against a physical model showing acceptable agreement (Thorenz, 2017). Thus, here it was the plan to validate the hydraulics and to determine predicted ship forces numerically. In order to narrow down the range of possible air contents, data from the prototype lock Bolzum was used. As the general hydraulic behaviour of the developed model was already validated by physical model tests of a different set-up (Schulze, 2018), a combination of prototype and numerical data was used to determine the inlet air content for the numerical model. Flow rates were computed from the measured water level development of the prototype. These flow rates were then prescribed as a boundary condition in the numerical model. Thus, the remaining free parameter was the air content of the inflow. As the air entrainment has a very significant impact on the surface flow velocities, video recordings of the water surface of the prototype were used to derive surface flow velocities. The comparison between the measured prototype velocities and the numerically computed velocities was used to calibrate the air content at the inlet of the numerical model. This lead to an air content of 10 % to 20 % in the inlet jet which can be used together with Figure 4 to draw conclusions about the allowable flow rates while filling the lock chamber.

5 Conclusions and future work

The new modelling approach allowed capturing the effect of the entrained air bubbles on the ship forces during the filling of a navigation lock. From the results it can be concluded that the entrained air significantly influences the flow regime in the front part of the lock. Due to buoyancy of the bubbles, a vertical velocity component is introduced into the air-water intermixing regions. As a consequence a jet evolves which is directed

Schulze, Stamm, Thorenz: A new two-phase flow model for the investigation of the effect of entrained air in navigation locks.
In: IAHR: E-proceedings of the 38th IAHR World Congress, September 1-6, 2019, Panama City, Panama. Madrid: 2019, S. 5763-5771.

S. 5771

towards the ship bow. When the jet reaches the hull, significant forces towards the downstream water develop. The forces can be limited by reducing the inflow discharge.

For quantifying the forces occurring at the prototype structure, measurements of the air content are essential. Alternatively, an entrainment model for the numerical approximation of the air entrainment through the plunging jet has to be developed and implemented.

As a rough approximation the falling jet could be discretized with a very fine resolution in combination with the usage of a Large Eddy Simulation (LES) turbulence approach. With this, the larger air pockets, which are produced by the plunging jet, can be directly captured. In combination with the new approach, the air entrainment transport and degassing processes are assumed to be captured with suitable accuracy for engineering purposes.

In future work, we are aiming to improve the capturing of the physical effects of the air entrainment and transport processes. Additionally, the collection of measurement data for comparison of the results is pursued.

6 References

Belzner, F., Simons, F., and Thorenz, C. (2018). An application-oriented model for lock filling processes, PIANC- World Congress 2018, Panama City.

Damian, S.M. (2013). An Extended Mixture Model for the Simultaneous Treatment of Short and Long Scale Interfaces. Doctoral Thesis. Universidad Nacional Del Litoral, Santa Fe, Argentinia.

Issa R. I. (1985). Solution of the Implicitly Discretized Fluid Flow Equations by Operator-Splitting, Journal of Computational Physics, 62, pp 40-65, 1985.

Jasak H. (1996). Error Analysis and Estimation for Finite Volume Method with Applications to Fluid Flows, Doctoral Thesis, Imperial College, University of London.

Menter, F. R. (1993): Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows, AIAA Paper 93-2906. Partenscky, H.-W. (1986). Binnenverkehrswasserbau: Schleusenanlagen. Springer Berlin Heidelberg, Berlin, Heidelberg.

Patankar, S. V., and Spalding, D. B. (1972). A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows, Int. J. Heat Mass Trans., 5(1972), p.1787–1806.

Rusche, H. (2002).Computational fluid dynamics of dispersed two-phase flows at high-phase fractions Phd thesis, Imperial College, London.

Schulze, L. (2018). Development of an Application-Oriented Approach for Two-Phase Modelling in Hydraulic Engineering. Dresden: Technische Universität Dresden, Institut für Wasserbau und technische Hydromechanik (Dresdner Wasserbauliche Mitteilungen, 61).

Thorenz, C. (2017). Numerical investigation of salinity effects in locks, presentation, 8th International PIANC- SMART Rivers Conference, September 18-21, 2017, Pittsburgh, USA, online available on 2019-06-17 at https://web.archive.org/web/20190617152106/ https://pianc.us/wp-content/uploads/2016/09/4- thorenz.numerical-investigation.pptx

Thorenz, C., Belzner, F., Hartung, T., and Schulze, L. (2017): Numerische Methoden zur Simulation von Schleusenfüllprozessen. In BAWMitteilungen Nr. 100: Kompetenz für die Wasserstraßen – Heute und in Zukunft Forschungs- und Entwicklungsprojekte der BAW. Bundesanstalt für Wasserbau, Karlsruhe, 91–108. Zheng, L., and Yapa, P.D. (2000). Buoyant velocity of spherical and nonspherical bubbles/droplets. Journal of Hydraulic Engineering 126(11), 852–854.

Autoren

Lydia Schulze
Federal Waterways Engineering and Research Institute, Karlsruhe, Germany
lydia.schulze@baw.de

Jürgen Stamm
Technische Universität Dresden, Dresden, Germany

Carsten Thorenz
Federal Waterways Engineering and Research Institute, Karlsruhe, Germany
carsten.thorenz@baw.de