Conference Paper, Author's Postprint

Thorenz, Carsten; Strybny, Jann

On the numerical modelling of filling-emptying system for locks

ON THE NUMERICAL MODELLING OF FILLING-EMPTYING SYSTEMS FOR LOCKS

CARSTEN THORENZ (1), JANN STRYBNY (2)
(1): Federal Waterways Research Institute (BAW), Kußmaulstr. 17, 76187 Karlsruhe, Germany
(2): Faculty of Maritime Sciences, University of Applied Sciences Emden/Leer, Bergmannstr. 36, 26789 Leer, Germany

This paper concerns a methodology for examining high lift locks with numerical tools. As an example the main features and initial studies of the filling-emptying system for a high-lift lock with water saving basins are presented. The hydraulic layout and the numerical models used for evaluation of the lock performance are described.

BACKGROUND

In the last years, several locks in Germany have been built with a filling-emptying system which is based on a pressure chamber beneath the lock chamber for the fluid distribution. This system was recently described in PIANC Report 106 “Innovations in navigation lock design” [1]. For canal locks in Germany, typically additional water-saving basins are attached to the filling-emptying system. Recently the Federal Waterways Research Institute (BAW) started to work on the hydraulic design for a new set of locks for the Main-Danube Canal. For this purpose the pressure chamber filling-emptying system with attached water saving basins was used. The system is similar, though larger, to the one planned and tested for the new lock of Minden (at the crossing of the Mittelland Canal over the river Weser, currently under construction). Figure 1 shows the general layout of the system.

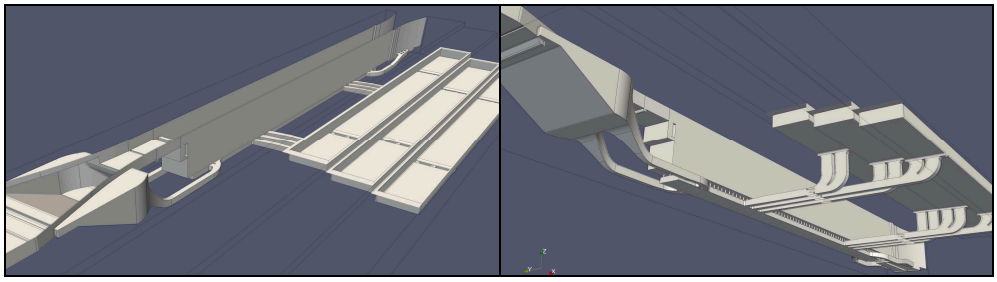

Figure 1. 3D Sketch of the proposed filling-emptying system seen from above and below

The upstream reach is on the left and connects to the pressure chamber below the lock chamber with two culverts. The attached water saving basins are on the right hand side of the drawing and connect to the pressure chamber, too. The pressure chamber itself is connected to the lock chamber by several hundred nozzles.

EVALUATION OF THE LOCK FILLING PROCESS

In the dimensioning process for navigation locks, two main constraints govern the design of the hydraulic system: The requested filling-emptying time for the lock and the allowed forces acting on the vessel. Thus, for locks of lower importance (i.e. with low traffic) a simple hydraulic system, which would lead to inacceptable ship forces if operated fast, can be acceptable for a slow lock operation. For locks of higher importance (i.e. traffic density and/or construction costs) it is necessary to evaluate the generated forces thoroughly in advance, in order to achieve a sufficient speed while keeping the hawser forces below the allowable limits. Traditionally, the fulfillment of this criterion would be tested with a laboratory model. Presently, numerical models are used in conjunction with or as a replacement for laboratory models. Here we describe the numerical methods to evaluate the general hydraulic behaviour of the lock and in detail the way to evaluate the sloshing of water in the lock chamber. Sloshing is one of the main parameters for the design, as it must be minimized for the sake of safety and comfort of ships.

NUMERICAL MODELLING OF THE FILLING-EMPTYING SYSTEM

Introduction

Numerical models have gained a lot of impact on the design process. In the BAW, one-dimensional network models are used in conjunction with the results from three-dimensional models and laboratory models for the early design stages and for later detail optimizations. Especially the three-dimensional modeling of the very small water surface differences in the lock chamber in conjunction with the large vertical displacement of the water surface during the locking process requires a remarkably high accuracy of the solver. The open-source package “OpenFOAM” was adapted and used for this task.

Coupling of numerical models

Currently it is not feasible to do design work with a three-dimensional numerical model for the whole locking process. The different hydraulically relevant scales range from a few decimeters (filling nozzles in the chamber floor) to several hundred meters (lock chamber with approach areas), making it difficult to capture all relevant physics with a single model. Thus, it was decided to regard different subsystems separately with different types of models in three steps:

1. detailed local three-dimensional single-phase models for relevant parts of the system (inlet, nozzles, etc.) and parameterization of the results
2. one-dimensional network model for overall dynamics and fluid balances (with parameterization from local three-dimensional models)
3. detailed three-dimensional multi-phase model for the lock chamber (with boundary conditions from one-dimensional model)

The combination of constrained space between ship and lock chamber (~0.5 m) with large lift height (~20 m) makes computations cumbersome, due to the necessity to morph the grid during the computation. In the future, we plan to circumvent this by modeling the ship with the "Immersed Boundary Method." Currently the filling process without the actual ship in the lock chamber is regarded instead.

One-dimensional numerical model

For the global dynamics of the system, the commercial code "Flowmaster" was used. It is based on the theory provided by Miller [5] concerning internal flows in one-dimensional networks. It relies heavily on parameterization, thus it was necessary to calibrate the local losses of special parts, for example the filling through the inlet, the ducts and the filling nozzles. For the case regarded here, only the filling of the lock from the upstream reach was tested with a simplified network model (Figure 2). For the further detailed design work, a more complex network was used, which includes the saving basins and the downstream reach.

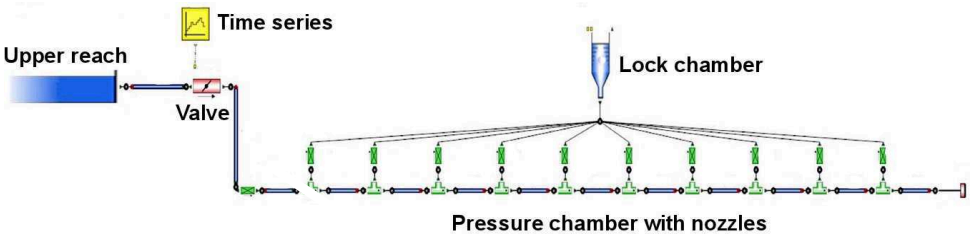

Figure 2. Network used for one-dimensional simulations of filling from the upstream reach

For this system, the nozzles were simulated by a T-piece combined with a concentrated hydraulic loss. The parameterization of this combination showed a sufficient accuracy compared to the laboratory model tests for the lock of Minden. For the inlet combined with the adjacent ducts, a new geometric element and several other parts, three-dimensional model runs were performed in order to obtain hydraulic loss coefficients. After calibration, the one-dimensional system was used to compute the dynamic behaviour of the total system (i.e. during the filling process).

Three-dimensional numerical multiphase flow model

As three-dimensional numerical code OpenFOAM was used. OpenFOAM is a library of numerical methods, based on a finite volume scheme in space with a finite differences discretization in time, developed by Henry Weller, Hrovje Jasak [2] and several collaborators. Here, the "interFoam" solver was used as a basis for further developments. "interFoam" solves the multiphase Navier-Stokes-equations with a Volume-of-Fluid (VOF) approach (Hirth et al. [6]). Details of the implementation can be found in references [3] and [4]. The implementation of interFoam changed quite significantly in the different versions of OpenFOAM: for all versions except OpenFOAM-1.6, variable replacement was performed for

the pressure in the equations. The physical pressure p was reduced by an approximation for the hydrostatic pressure:

$$p_{rgh} = p - \rho \cdot (\vec{g} \cdot \vec{z}) \tag{1}$$

Here p_{rgh} is the newly introduced primary variable, p the physical pressure, ρ the fluid mixture density from the VOF approach, $\vec{g}$ the vector of gravity acceleration and $\vec{z}$ the coordinate vector of the regarded point. This approach has the following advantage: if we imagine a resting fluid with its free surface located at $\vec{z} = [0 \quad 0 \quad 0]$, we can easily see that the correct solution for p would be $p = \rho \cdot (\vec{g} \cdot \vec{z})$ (assuming p=0 at the surface), thus the hydrostatic gradient in p is perfectly canceled in our new primary variable p_{rgh}. In this simple case, p_{rgh} is equal to zero everywhere in the domain. From a numerical point of view, it is very favourable to cancel gravity acceleration term and hydrostatic pressure gradient before discretizing the equations, as it is difficult (if not impossible on unstructured grids) to achieve a perfect cancellation after the terms have been discretized. Furthermore, iterative solving of the resulting linear equations system for p_{rgh} is trivial in this case.

Unfortunately, difficulties arise on other fronts. If we imagine a resting fluid with its free surface far away from $\vec{z} = [0 \quad 0 \quad 0]$, we can easily see that we will have a sudden change in the newly introduced primary variable p_{rgh}. This is due to the fact that the density ρ switches at the interface between the fluids and thus the product $\rho \cdot (\vec{g} \cdot \vec{z})$ switches, too. Due to this jump, the new primary variable p_{rgh} is no longer continuously differentiable; the gradient is equal to infinity at the interface. This problem may be acceptable if the distance of the free interface from the coordinate origin is small (simulation of a ship in the ocean), but it grows with increasing distance. Our experience has shown that the numerical results are not invariant against translation of the model in space. This is significant if the distance is much larger than the fluid depth (e.g. simulation of a lock at the Main-Danube-Canal, 300 m above sea level).

Thus, we decided to base all developments on the direct discretization with p as primary variable as it is implemented in OpenFOAM-1.6. This has the additional advantage that the formulation of boundary conditions is much easier from an engineer's point of view, as we are applying them to a physically relevant quantity. Unfortunately, it has the disadvantage that the pressure gradient and the gravity acceleration term do not cancel easily anymore and as a result spurious velocities can occur in areas where the mesh is distorted. To circumvent this problem, choosing a fourth-order scheme for the pressure field calculations in OpenFOAM provided the best results so far. However, due to the density jump, errors are introduced directly above the water surface in the air, leading to unphysical movement of the air. At the time given, we assume that this is due to the kink in the pressure profile (second derivative equals infinity) in combination with the fourth order scheme for the pressure. For our class of problems (simulation of hydraulic engineering problems) the quality of the simulation results are most relevant for the dense fluid (water) but less important for the light fluid (air). Therefore, the issue pointed out above is of minor importance for us.

Implemented Boundary Conditions

In hydraulic engineering practice, simplified and usable boundary conditions are necessary. The effort to set up a model must be reduced to a minimum. We prepared a set of boundary conditions which are relevant for hydraulic engineers. Widely applied models are driven either by a fixed inflow on the upstream and a fixed water level at the downstream or by different water levels. During the work with OpenFOAM, it became clear that the object-oriented implementation of OpenFOAM greatly simplifies the implementation of complex boundary conditions, though the first steps are rather hard.

Fixed water table

Implementing a robust scheme for a fixed water table had a high priority. We chose an approach where the water table is prescribed via the pressure field for each cell on the boundary:

$$p_{wt} = p_0 - \beta\rho(\vec{n}\cdot\vec{v})^2 + \begin{cases} \rho_{light}(\vec{g}\cdot(\vec{z}-\vec{z}_0)) & \text{if} \quad \vec{g}\cdot(\vec{z}-\vec{z}_0)<0 \\ \rho_{dense}(\vec{g}\cdot(\vec{z}-\vec{z}_0)) & \text{if} \quad \vec{g}\cdot(\vec{z}-\vec{z}_0)>0 \end{cases}$$
$$\text{with} \quad \beta = 0 \quad \text{if} \quad \vec{n}\cdot\vec{v} > 0 \tag{2}$$

In these equations, p_{wt} is the prescribed pressure at the boundary, p_0 is the pressure at the prescribed water surface, z_0 is a point on the prescribed water table, n is the face normal vector of the boundary patch, $\vec{v}$ is the velocity vector. With the parameter β we can adjust a stabilization for inbound flows, at the expense of slightly misadjusting the pressure. Furthermore, the VOF function α (which denotes the degree of filling of cells with water) is prescribed at the boundary:

$$\begin{aligned} &\text{if} \quad \vec{n}\cdot\vec{v} > 0: && grad(\alpha) = 0 \\ &\text{if} \quad \vec{n}\cdot\vec{v} < 0: && \alpha = \begin{cases} 0 & \text{if} \quad \vec{g}\cdot(\vec{z}-\vec{z}_0)<0 \\ 1 & \text{if} \quad \vec{g}\cdot(\vec{z}-\vec{z}_0)>0 \end{cases} \end{aligned} \tag{3}$$

The gradient of the velocity vector is set to zero at the boundary. For all other transported quantities (i.e. turbulence model equations) a gradient equal to zero is prescribed for outbound flows and a constant Dirichlet boundary condition is prescribed for inbound flows.

Fixed flowrate of water

Furthermore, several inlet boundary conditions for the velocity field were implemented. Most relevant from the engineer's point of view is a velocity boundary condition for a fixed volume flux of the dense phase, independent from the fluid phase distribution (i.e. the water table):

$$\begin{aligned} &\text{if} \quad Q > 0: && \vec{v} = \vec{\mathrm{n}} Q / \int_{\Omega} \alpha \, d\Omega \\ &\text{if} \quad Q < 0: && \vec{v}_{boundary} = \vec{\mathrm{n}} Q / \int_{\Omega} \alpha \, d\Omega + \left(\vec{v}_{domain} - \left(\vec{v}_{domain} \cdot \vec{\mathrm{n}} \right) \cdot \vec{\mathrm{n}} \right) \end{aligned} \tag{4}$$

In these equations Q is the prescribed flowrate and Ω is the regarded boundary. For outbound flow ($Q<0$) special treatment is necessary in order to transport the transversal parts of the velocity out of the domain. Otherwise, an unphysical accumulation of transversal momentum would occur on the boundary.

VALIDATION OF THE NUMERICAL MODELS

For the lock at Minden, both numerical and laboratory model tests were performed in the BAW during the hydraulic design of the lock. A laboratory model (scale 1:25, based on Froude similarity) was used in an hybrid modeling approach together with numerical models. Parameters obtained from the laboratory model were used for the numerical models and the numerical models were used to pre-evaluate design alternatives before building them in the laboratory model. In the laboratory model, the ship is partially fixed by a measurement unit. The ship has been given three degrees of freedom: Vertical lift, rolling around its longitudinal axis and rolling around its transversal axis. The movement in the horizontal plane (x-y plane) was not possible and the necessary forces to keep the ship in position were recorded. Rotation around the vertical axis was also prevented and the moments were recorded. The results from this laboratory model where used for validation of the numerical tools.

For the validation of the code, several tests were performed. The water slope in the lock chamber is most relevant for the locking process. It is difficult to evaluate the position from the VOF function with sufficient accuracy, because the shape of the isosurface for α=0.5 (i.e. the water surface) is often disturbed from interpolation errors on unstructured grids. Thus, we decided to mimic the methodology from laboratory model tests. Under the assumption of a hydrostatic pressure profile in the chamber (accurate enough far away from the nozzle jets) the position of the water table was computed from the pressure field. In order to avoid post-processing errors, this was realized in the computational code itself by introducing a new field variable WSP which computes the position of the water table from the pressure field in the dense phase:

$$WSP = \left| \vec{z} - \frac{\vec{g}}{|\vec{g}|} \frac{p - p_0}{\rho \cdot |\vec{g}|} \right| \tag{5}$$

The gradient of WSP multiplied with the ships mass force $|\vec{g}| \cdot m$ gives a good approximation of the longitudinal forces acting on a ship in the chamber [1]. This has been done with a two point discretization, one point at each end of the lock chamber. The results

are presented in Figure 3 for one of the test runs performed with the model for the lock of Minden together with the results computed from the simulation. The results show an excellent agreement for the computed forces for the first peaks with a negligible phase deviation later. For this test, the sloshing without a ship in the chamber was regarded, as the simulation of the moving ship in the chamber was not possible yet.

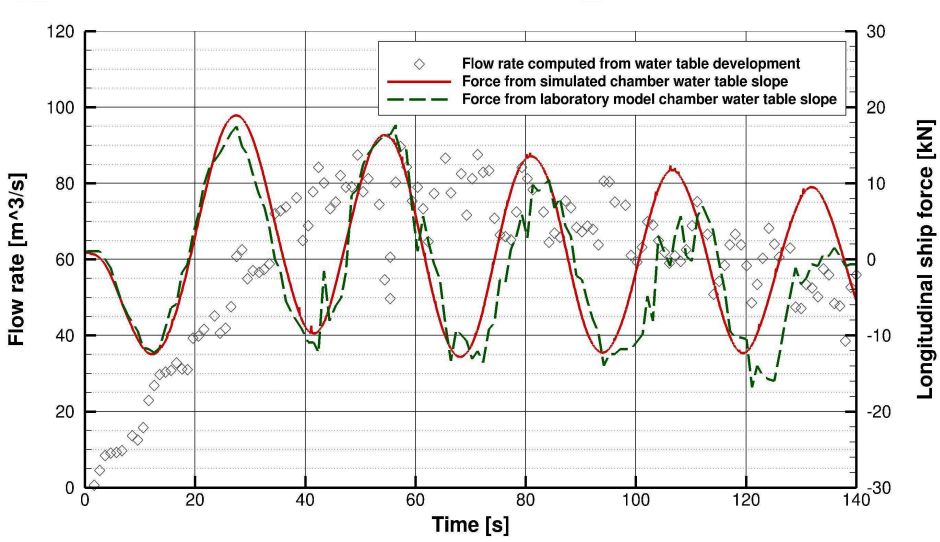

Figure 3. Ship force calculated from laboratory model test water table (green, dashed) and simulation results (red, solid). The circles represent the flow rate to the lock chamber.

After this successful validation, the three-dimensional simulations for the new lock were performed. We used the same grid resolution (6 Mio. cells), which is just acceptable in order to reproduce the flow separation at the nozzles (Figure 4, right). Though the filling process starts from the upstream end (Figure 4, left), the resulting water table slopes in the chamber were acceptable.

Apart from the filling process from upstream, several other processes were regarded: the development of vortices and the head losses at the intake, filling from the water saving basins, wave formation in the water saving basins and the efficiency of the downstream stilling basin were tested with three-dimensional sub-models. All of them were driven from the flow rates delivered by the one-dimensional model and the results for the local head losses were fed back into the one-dimensional model.

All OpenFOAM computations where performed on an SGI Altix ICE 8400. For one testcase ($6 \cdot 10^6$ cells) the parallel speedup was evaluated. We observed a speedup of 3.6 when moving from 24 to 96 cores. Further decomposition from 96 to 192 cores resulted in a speedup of 1.8, even further increase in the number of cores did not achieve a sufficient speedup anymore.

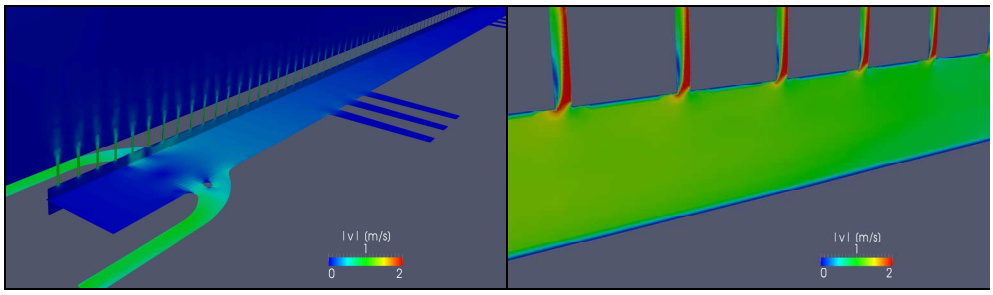

Figure 4. Velocities while filling the lock in horizontal and vertical cross-section of the filling system. Detail of the nozzles at the right.

SUMMARY

The combination of one- and three-dimensional models enables the simulation of lock filling processes with sufficient accuracy. The open source package OpenFOAM was used in conjunction with the commercial code Flowmaster for this task and enhanced for hydraulic engineering usage by implementing additional boundary conditions and enhancements to the solver. Validation test cases have shown an excellent agreement to the predicted water table oscillations in the lock chamber with the results from laboratory model tests. The validated code was successfully applied in several studies for a new lock.

The developed source codes can be downloaded after the conference from this address: http://www.baw.de/methoden/index.php5/OpenFOAM. A number of further test cases will be found in the additional paper "On the numerical modeling of ship-waterway-interaction in canals" by *Strybny et al.* at this conference.

REFERENCES

[1] PIANC, *"Innovations in Navigation Lock design"*, PIANC, Brussels, (2009), pp. 76-77.

[2] Weller, H.G.; Tabor G.; Jasak, H. and Furby, C., "A Tensorial Approach to CFD using Object Orientated Techniques", *Computers in Physics*, Vol. 12 No. 6, (1998), pp. 620–631.

[3] Ubbink O., *"Numerical prediction of two fluid systems with sharp interfaces"*, PHD thesis, Imperial College London, (1997).

[4] Rusche H., *"Computational fluid dynamics of dispersed two-phase flows at high phase fractions"*, PHD thesis, Imperial College London, (2002).

[5] Miller D. S., *"Internal Flow Systems"*, British Hydromechanics Research Association, 1978.

[6] Hirt, C.W. and Nichols, B.D., "Volume of fluid (VOF) method for the dynamics of free boundaries", *Journal of Computational Physics,* Vol. 39, No. 1, (1981), pp. 201–225.