Conference Paper, Published Version

Schulze, Lydia; Rusche, Henrik; Thorenz, Carsten

Development of a Simulation Procedure for the 3D Modelling of the Filling Process in a Ship Lock Including Fluid Structure Interaction

Paper 10 – Development of a Simulation Procedure for the 3D Modelling of the Filling Process in a Ship Lock Including Fluid Structure Interaction

SCHULZE L., RUSCHE H., THORENZ C.;
Bundesanstalt für Wasserbau (BAW; Federal Waterways Research Institute), Karlsruhe, Germany ; WIKKI GmbH, Braunschweig Germany; Bundesanstalt für Wasserbau (BAW; Federal Waterways Research Institute), Karlsruhe, Germany ;

Email (1st author): lydia.schulze@baw.de

ABSTRACT: The analysis of the forces on a ship during the lockage with the means of numerical modelling is a big challenge. The modelling of the lifted ship requires special methods for performing the fluid structure interaction. In particular, large lifting heights and small spaces between the ship and the lock chamber walls make a simulation of the locking process including the lifted ship very difficult. A possible modelling concept for this particular application will be described in the scope of this paper. Further, practicability and the results will be evaluated.

1 INTRODUCTION

A good compromise between a fast and safe filling and emptying procedure on the one hand and small construction and maintenance costs on the other hand is the ultimate goal when designing a ship lock. For the filling and emptying the forces acting on the ship hawsers are the critical design criteria. The forces arise due to movements of the water in the lock chamber during the filling and emptying process. When the same valve speed is used for filling and for emptying, the forces acting on the ship are higher during the filling, since the water cushion underneath the ship is smaller (Partenscky 1986). Therefore, the following study only considers the filling process.

For lifting heights smaller than 10 meters the locks are usually filled directly through the upstream gate or through valves in the gate. In these systems, the filling process causes waves in the longitudinal direction of the lock chamber which produce strong longitudinal forces acting on the ship hawsers.

For lifting heights taller than 10 meters special filling systems with lateral culverts on the sides of the chamber or a pressure chamber underneath the actual lock chamber are used for enabling a more balanced filling. However, most systems are not completely balanced and tend to fill the chamber faster on the upstream side. As with the before mentioned systems for smaller lifting heights, the filling process also causes a longitudinal incline in the water level surface in the lock chamber resulting in longitudinal forces on the ship hull. In Germany, those forces acting on a ship in an inland waterway lock during the filling should not exceed a maximum of 1/600 times the gross ship weight (Partenscky 1986). Values for ship locks in other countries differ slightly, more detailed information can be found in the latest PIANC report of Working Group 155 (2015).

The determination of those forces can be done by applying physical scale models. For that complex measurement structures are installed on top of the ship for analysing the acting longitudinal and transversal forces resulting from the tilting of the water surface in the ship lock chamber during the filling. However, high construction costs, long construction times, the restriction to very few measuring points and especially unavoidable scale effects make the physical models disadvantageous in some cases and strengthen the need for alternative investigation methods like numerical models.

Yet, the analysis of the forces on the ship with the means of numerical modelling is still a big challenge since the modelling of the lifting ship requires special methods. In particular, large lifting heights and small spaces between the ship and the lock chamber walls make a simulation of the locking process including the lifted ship very difficult. For the simulation special fluid structure interaction methods have to be applied.

In the scope of this paper, a possible modelling concept for this particular application is described. Further, practicability of the used method chain and the results will be evaluated.

2 FLUİD STRUCTURE INTERACTİON

With the methods of fluid structure interaction (FSI) the motion of solid bodies due to internal or surrounding fluid flow can be modelled. For that certain data has to be transferred between the fluid flow and the structural field. Therefore the fluid flow equations and structural equations are solved either in an iterative system or with a fully coupled system (Sieber 2002).

The iterative system starts with solving the fluid equations and passes the resulting pressure and shear forces to the structural motion solver. The forces are subsequently used to calculate the resulting motion of the movable rigid bodies. Next, the grid of the fluid domain is adapted and the old results are mapped to the new mesh. Then, the fluid flow equations can be calculated again based on the new mesh. The steps can be repeated once or several times until the next time-step is started. For fully coupled systems both equation systems are solved simultaneously in one combined system of equations using a monolithic scheme.

The iterative system for FSI has the advantage that specialized methods for both solvers can be used. A fully coupled system requires special software that solves both fluid flow and body motion in one large linear equation system. A drawback of the iterative method is the time lag between the fluid and the structure solution, which results in the need for relatively small time-steps. Further the explicit coupling might result in convergence problems. However, the possibility to use existent, validated and specially adapted solution methods for each solver and thereby achieving high efficiency and accuracy is the main advantage of iterative fluid structure methods.

In this study an iterative coupling method was used, which uses an adapted version of the fluid flow solver interFoam in combination with a motion solver both included in the Open Source package OpenFOAM. The numerical solution procedure of the solver used is explained below.

The simulation of a movable ship hull floating on a free water surface is one of the classical application examples in the field of fluid structure interaction. In most cases the ship is floating on open water and the motion resulting from waves or the ship movement is investigated. In the scope of this study, the filling process of a lock with a lifting ship in the chamber is investigated. This case is especially challenging, since the ship performs a large vertical movement within the narrow lock chamber.

3 DYNAMİC MESH SİMULATİONS WİTH OPENFOAM

With the Open Source C++ library OpenFOAM, two general forms of dynamic mesh simulations can be created: Simulations with prescribed motion or simulation with solution-dependent motion of the boundary. The first form is used when the motion of a boundary is known and can be described with parameters. A widely known application where prescribed motion is used is the simulation of rotating turbines or the opening of valves.

When the motion is not known beforehand but is caused by fluid flow, the solution dependent motion is assigned to certain boundaries. This can for example be used to simulate floating objects or wind-induced deformation of structures. To enable the dynamic mesh simulations at least one of the following three methods has to be included (Jasak, Tukovic 2010):

- Automatic Mesh Motion (mesh deformation)
- Layer Addition and/or Removal (topology change)
- Sliding Interface (interpolation between non-conform cell faces).

Automatic mesh motion implies that the vertices of the mesh are displaced during the simulation. The displacement is based on the motion of the boundaries of the movable objects. Mesh topology is preserved, only the coordinates of the vertices are changed. Through the point shifting the mesh is deformed. This method is suitable for small, smooth movements.

Rapid large movements result in a distorted mesh and unsatisfying mesh quality. For that, the layer addition and removal method can be additionally applied. Based on maximum and minimum layer thickness, cell layer can be created or removed in a predefined region. By this, the mesh topology is changed and results have to be mapped from the old to the new mesh.

When a complete mesh region has to be moved along an interface, a sliding interface can be created. The sliding interface enables the movement of the cell zones along the interface; fluxes between the non-conform cell faces are interpolated. The mesh topology in the cell regions is conserved.

4. CONCEPTİON FOR MODELLİNG A SHİP LOCK FİLLİNG PROCESS WİTH MOVABLE SHİP HULL

For the modelling of the filling process of the ship lock the Open Source CFD-toolbox OpenFOAM-extend was used. This toolbox is a C++ library designed for the calculation of partial differential equation discretized with three-dimensional polyhedral meshes. In addition to the standard toolbox customized tools were developed and applied for enabling the described simulations. All simulations were performed in parallel on the in-house high performance computer. Post processing was done with ParaView.

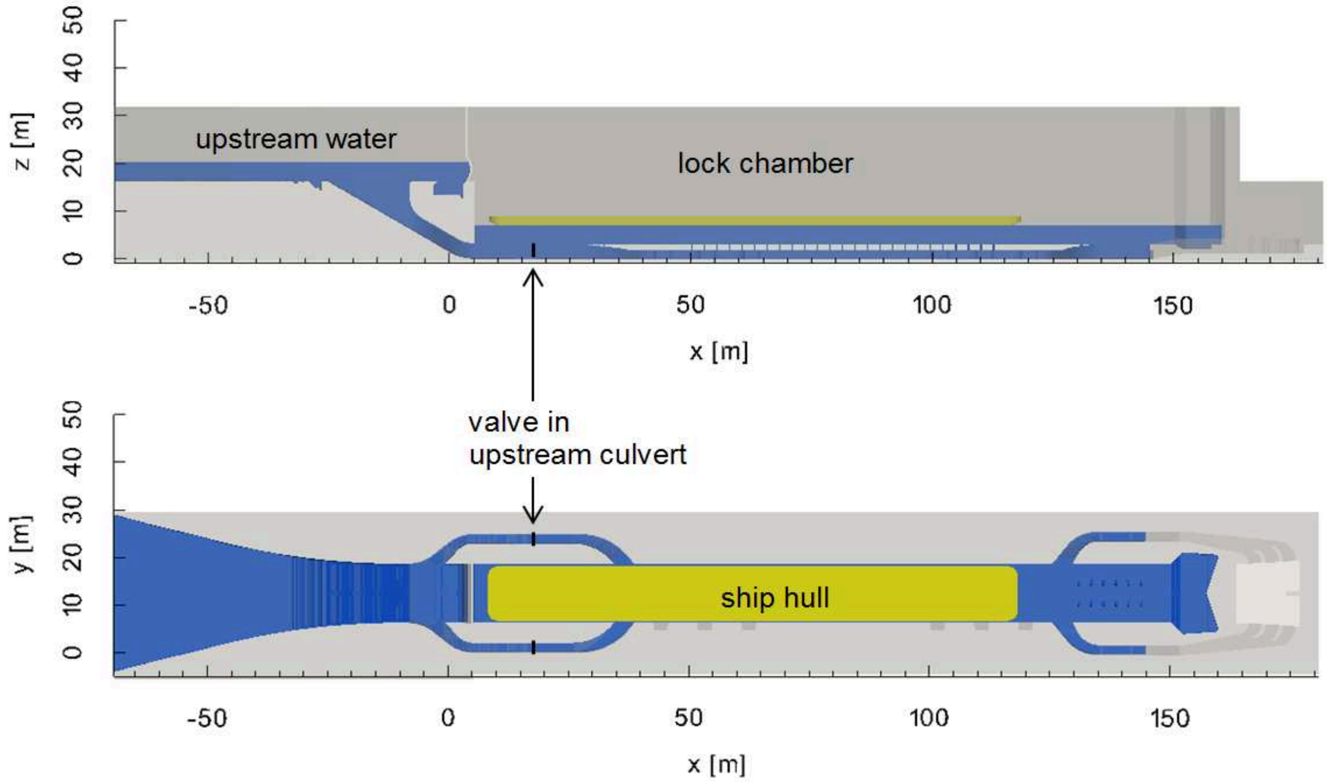

Figure 1: Sketches of the investigation subject: Upper Sketch: Longitudinal section through the lock, Lower Sketch: Top view on the lock

4.1. Grid Generation and Manipulation

The generation of the mesh for the simulations was performed with the OpenFOAM grid generator tools *blockMesh* and *snappyHexMesh*. With the first tool, a block-structured Cartesian grid consisting of 13 blocks was created. This setup was necessary for ensuring that planes of faces are generated for layer addition and removal as well as the generalized grid interface used to decouple the static and moving parts of the mesh close to the top gate.

Based on the structured blockMesh, the geometry adapted polyhedral mesh was created with the *snappyHexMesh* tool. In two steps (*castellatedMesh, snap*) the cells of the blockMesh are refined, deleted where external geometry is intersecting, and adapted to the geometry.

In a next step the mesh had to be prepared for the latter creation of internal interfaces, used for the mesh manipulation during the simulation. In particular a plane vertical and a plane horizontal layer of cell faces had to be created on pre-defined locations. For that the cell faces in a certain region in proximity to the latter planes were filtered with the *setSet* utility, which collects faces based on various criteria and writes them into a separated list. The sets can then be reintegrated into the actual mesh definition by translating the set into a zone with the *setsToZones* utility. For the created face zones the average x-coordinate or z-coordinate were calculated, then the filtered cell faces were moved to this location. This adjustment procedure for the cell faces was performed with the developed extension *adjustLockMesh*.

Last, an additional set of faces for the simulation of the filling valve in the lateral culvert was filtered. The set was also transformed into a zone for enabling the time adapted permeability of the faces.

4.2 Definition of Movable and Non-Movable Zones

The mesh for the lock simulation consists of a movable and a non-movable cell zone. The movable zone includes the main parts of the ship lock chamber. The upstream water and the pressure chamber underneath the lock chamber belong to the static region. Figure 2 illustrates the allocation of the static and the dynamic zone.

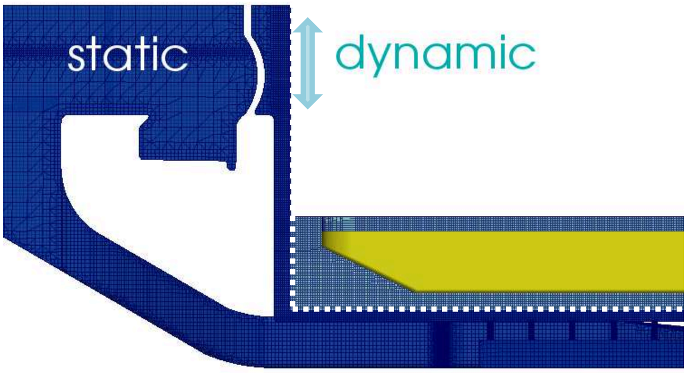

Figure 2: Dynamic and static zone for the numerical simulation of the lockage.

Both cell zones are connected by two interfaces: one vertical interface in front of the ship and another horizontal interface underneath the ship.

4.3. Definition of the interfaces between the zones

For the connection of the static and the dynamic mesh region two interfaces were defined. The first interface is located in front of the ship hull to enable the vertical sliding of the moving mesh region inside the lock chamber. It is defined as a generalized grid interface (GGI) and consists of two patches. Since the movable part slides upwards during the filling process, the conformity of the patches gets lost, meaning that normal flux calculation is not possible. Using the GGI definition, the coupling of the two non-conformal mesh regions is enabled. In particular, weighted interpolation is used to calculate the fluxes through partially overlapping faces (Beaudoin, Jasak 2008).

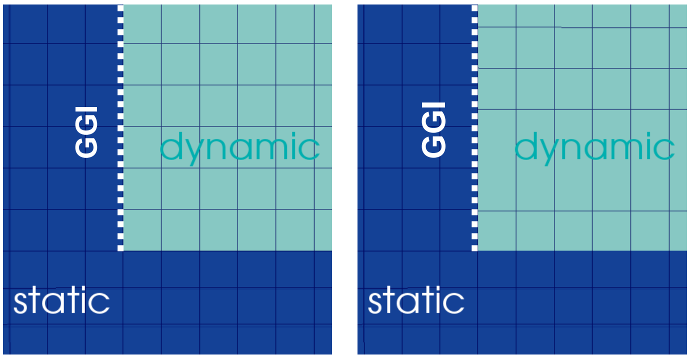

Figure 3: The mesh is dynamically adjusted during the simulation. Interpolation at the non-conform interface is performed with the GGI method.

For that, the neighbour relations are determined, meaning that an algorithm is used to check which faces of the master patch overlap with which faces on the slave patch. Then, the overlapping weights of the master and the cell face are calculated and all fluxes through the GGI are multiplied with the weights.

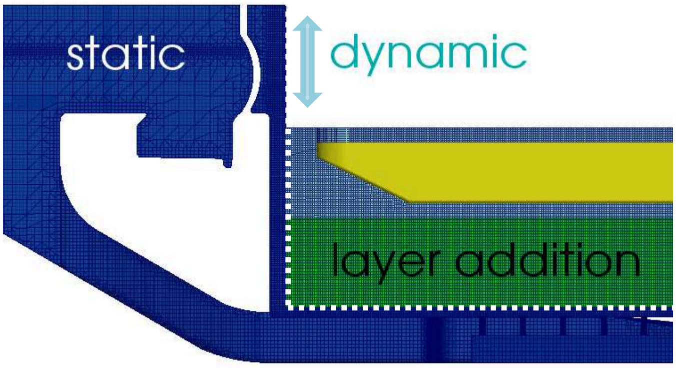

Figure 4: During the simulation layers are added underneath the ship hull.

The second interface at the bottom of the movable zone was constructed for defining the location for layer additions during the simulation. This is necessary because all cells in the movable region are allowed to be stretched only up to a certain thickness. When the maximum thickness is exceeded a new layer of cells is added. This procedure avoids the evolution of strongly stretched non-orthogonal cells in the layer addition zone which could increase the numerical error and could lead to instabilities.

4.4. Definition of the Rigid Body Motion of the Ship Hull

The hull of the ship describes a part of the boundary of the moving zone and is defined as rigid body with two degrees of freedom (DoF). When the ship is moved due to the interaction with the fluid, cells around the ship are deformed. The motion of the hull is calculated based on the 6 DoF equations, where in our case the movement of the hull is limited to two degrees of freedom namely the lifting (translational movement in z-direction) and the pitching (rotational movement around the lateral axis of the ship) are free, the remaining degrees are fixed. This was chosen to simplify the simulation.

4.5. Definition of the Valve Opening

For the operation of the lock filling the valves in the upstream culverts are opened. In the numerical simulation this valve opening is performed by manipulating the permeability of cell faces over time. In particular, a plane set of faces is selected and transformed into a zone. The flux through the selected cell faces is dependent on their height-coordinate (z-coordinate) and the time. Every time-step the coordinate of each cell is compared with a user-defined table which defines the opening height over time. When the centre of the cell face is below the calculated opening height, the flow can penetrate. For all cell faces above the calculated opening height the flux through the faces is prohibited. Figure 5 shows a section through the valve at two different time-steps.

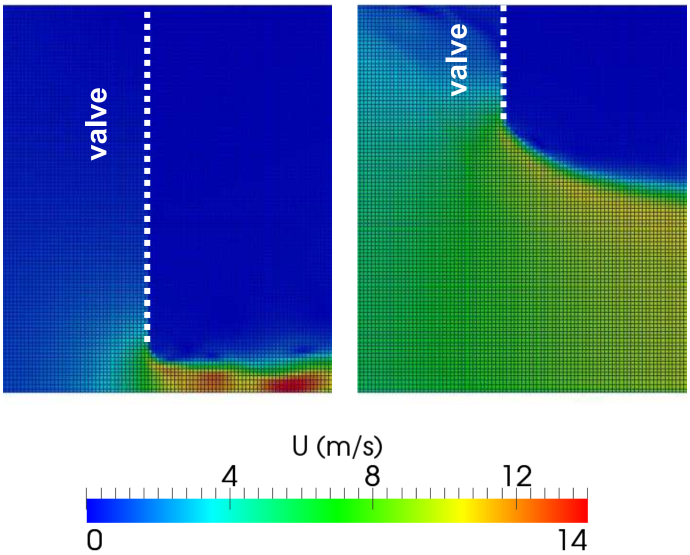

Figure 5: Velocity distribution of the flow beneath the simplified valve: In the valve plane the flow can only penetrate the faces below a calculated opening height. The opening height can vary over time.

With this method, a step-wise opening of the valve can be simulated in a simplified way without using dynamic mesh methods. A completely smooth opening is not possible, but a good approximation is achieved, when the cells in the region of the valves are refined.

5 THE NUMERICAL SOLUTİON PROCEDURE

For the simulation of the ship lifting process, the Finite Volume Method is used for the discretization of the fluid flow equations, which are solved with a segregated solution algorithm. The resulting viscous and pressure forces are integrated over the surface of the rigid body and the acceleration of the ship hull boundaries based on the rigid body properties like the moments of inertia, mass etc. is calculated. With the mesh motion solver the resulting velocities are transferred into mesh motions. The boundaries are displaced and the mesh is adapted correspondingly.

5.1. The Fluid Flow Equations

For modelling the flow of water and air in the simulation with focus on the prediction of the water surface in the chamber, the Volume of Fluid (VoF) approach was applied. This approach solves one mass and one momentum equation for the fluid mixture. The fluid mixture is assumed to be incompressible. Additionally, an advective transport equation is solved, which determines the volume fraction in each cell. Since this equation is likely to either smear the surface due to numerical diffusion. To counteracting this smearing high order discretization schemes are used and an additional term is introduced into the advection equation.

The set of equations used for the fluid flow simulation can be formulated as follows (Rusche 2002):

Mass conservation equation:

$$\nabla \cdot U = 0 \qquad (1.)$$

Momentum conservation equation:

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U U) = -\nabla p_{-rgh} + \nabla \cdot [\mu(\nabla U + \nabla U^T)] - g \cdot x \nabla \rho + F_{st} \qquad (2.)$$

Volume of Fluid equation:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha U) + \nabla \cdot ((1-\alpha) U_r \alpha) = 0 \qquad (3.)$$

Where

$$\rho = \alpha \rho_1 + \rho_2 (1-\alpha)$$
$$\mu = \alpha \mu_1 + \mu_2 (1-\alpha)$$

and

$$p_{-rgh} = p - \rho g \cdot x$$

ρ: density of the mixture
ρ_1: density of phase 1 (water)
ρ_2: density of phase 1 (air)
U: velocity of the mixture
p: pressure
μ: dynamic viscosity
μ_1: dynamic viscosity of phase 1 (water)
μ_2: dynamic viscosity of phase 2 (air)
g: gravity
x: position vector
F_{st}: surface tension
α: volume fraction
U_r: artificial compression velocity

The third term in the VoF equation acts only in the region close to the interface where the volume fraction ranges between $0 < \alpha < 1$. With the artificial velocity U_r being directed perpendicular to the water surface it artificially compresses the interface region. As the volume fraction is a bounded variable, getting non-physical out of its bounds, it is very important to conserve the boundedness. For that, a special solving algorithm including the Multidimensional Limiter for Explicit Solutions (MULES) was applied.

Within one time-step the volume fraction equation is solved first, based on the velocity values of the previous time-step. The new volume fractions are then used to calculate the mass and momentum equation. Since these are coupled, a pressure-velocity coupling algorithm is used. After the solution of mass and momentum, the turbulence equations are solved and the turbulent viscosity is corrected. In the scope of this study, the k-omega-SST two-equation model was applied.

5.2. Adaption of the Solver for Enabling Fluid Structure Interaction

The top-level solver implements the Finite Volume implementation of the Volume-of-Fluid free surface flow solver in transient formulation. It is derived from the naval hydro pack distributed by Wikki Ltd. and has support for dynamic mesh CFD simulations, including moving deforming mesh, topological changes and GGI. Fluid structure interaction is performed by two-way coupling: The fluid dynamic forces acting on the ship hull and the resulting motion by solving the 6-DoF Ordinary Differential Equation for a rigid body which is then imposed onto the computational mesh. Here, mesh motion is treated through a novel analytical mesh motion technique which dampens out the effects of rotation based on the relative distance to the lock walls. This approach reduces the computational overhead of the moving mesh quite substantially.

5.3 Boundary and Initial Conditions

On the upstream water boundary, a fixed water level was set to simulate the upstream water channel, which provides the water for the filling. For that, an in-house boundary condition formulation was applied, which ensures a water level which stays on the same level (Thorenz, Strybny 2012). The top of the domain was defined as open to the atmosphere with the relative pressure set to 0. All boundaries resulting from the geometry are defined as walls. For reducing calculation effort, only half of the lock was modelled, applying a symmetry boundary at the longitudinal side of the cut model.

Before starting the simulation, the ship lock was filled with water (volume fraction $\alpha = 1$) up to a water level of 4 metres in the lock chamber. The upstream water level was also set to 4 metres above the upstream bottom. The valve was completely closed and its opening velocity was defined via a lift curve describing the opening width over the time. For the latter comparison to the physical scale model, a constant opening speed of 0.01477 m/s (220 seconds for 3.25 metres valve height) was chosen. The height of the cells in the valve region was approximately 0.031 m so that the opening was performed in 104 steps.

6 RESULTS OF THE SHİP LOCK SİMULATİONS

6.1 Performance of the Simulation

The simulation of the ship lock filling was performed in parallel on the in-house high-performance computer using 16 CPUs on two physical nodes. The decomposition was performed with an adapted metis algorithm, which made sure that all faces of the GGI interface were located on one CPU. A total computation time of approximately 14 days (with several restarts) was needed to simulate 200 real time seconds.

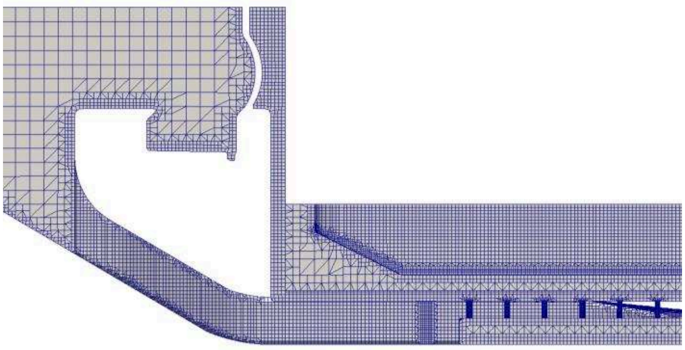

Figure 6: Mesh of first simulation at time-step 0 s.

Whereas the pre-processing of the mesh created lots of trouble, the used mesh adaptions during the simulation including the deformation of the movable zone and the insertion of new layers worked satisfactory (see Figure 6 and 7).

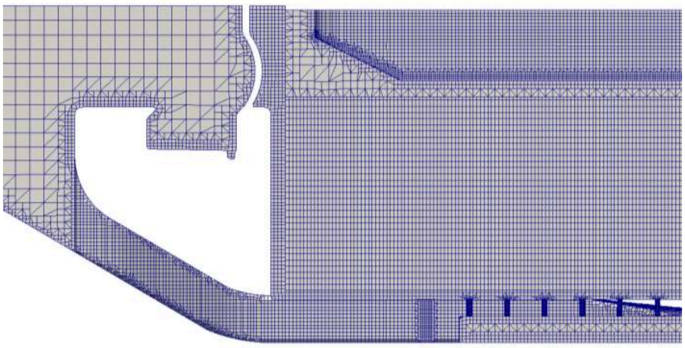

Figure 7: Mesh of first simulation at time-step 337 s.

6.2 Discussion of the Results

After optimizing the choice of discretisation schemes, the simulation was relatively stable. Still, the waves arose at the water surface possibly resulting from regions where the size of the grid cells changes. Further, un-physical high velocities in the air filled zone occurred. Such non-physical velocities in the air face are a widely known problem when applying the Volume of Fluid approach .

For evaluating the quality of the results of the numerical simulation, data of a physical scale model was used for comparison. In particular, the magnitude and the evolution of the forces acting on the ship were considered.

The physical model is a 1:25 scale model of the lock Bolzum, constructed to investigate the hydraulic behaviour of the planned lock. The model was build and data was collected in 2008. For the data collection, floating gauges for the water level determination and a construction for the force measurements on the ship were installed (Kemnitz 2008).

For the comparison with the numerical model three measuring cases with the necessary setup were available. Since the model is not existent anymore, new data collection was not possible. Figure 8 shows the comparison of the numerical and the physical model results concerning the longitudinal forces on the ship during the lock filling. Over the complete valve opening time of 230 seconds the numerical model shows smaller forces than the physical model. The first peak, indicating a pushing of the hull towards the downstream end, is reached already after 16 s in the numerical model whereas the physical model reaches the peak only after 21 s. The second peak is reached almost simultaneously in both models. The third peak is again delayed between the models, whereas the magnitude of the forces is in the same range.

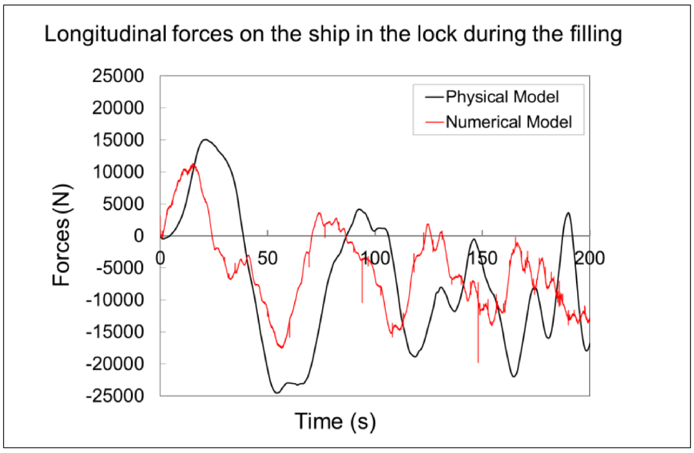

Figure 8: Plot of the longitudinal forces on the ship during the filling. Physical model measurements versus numerical simulation with FSI.

The differences between the physical and the numerical model can have various reasons:

One reason is assumed to be the grid resolution which was too coarse in some regions. In the first simulation, which was used for the forces plot, we used a coarse grid with large size gradients for testing the general modelling methodology. The filling nozzles, which are used to fill the lock chamber, had a resolution of 6 to 9 cells only over the diameter of 0.35 m (see Figure 9). This poor resolution might have caused wrong flow rates through the nozzles. Generally, a suitable grid resolution for the complex filling system is a challenging task, as features of very different scales have to be considered: whereas the pressure chamber has a length of approximately 200 m, the filling nozzles have a diameter of 0.35 m. Further results with better mesh quality are running currently and are still to be evaluated.

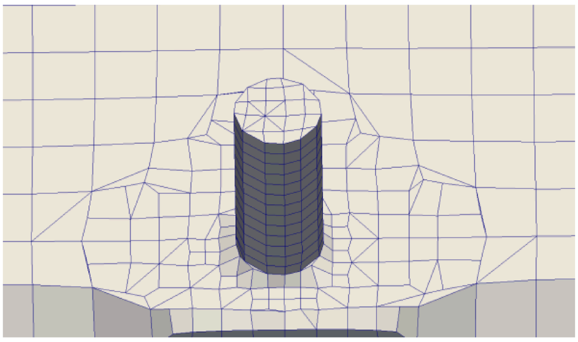

Figure 9: Coarse grid resolution in filling nozzles

Further, errors in the simulation may have resulted from the interpolation of the fluxes in the GGI region as well as from the discretization in the in the dynamic mesh region, where cells are deformed and the mesh topology is adjusted from time to time.

Naturally, differences can also result~~s~~ from different construction, functionality and boundary conditions between the two models:

- For example, the measuring system in the physical model uses different degrees of freedom for the ship. In particular, the ship has six degrees of freedom, where the translational motion in x- and y-direction and the rotational around the z-axis is restricted by springs. Forces on the hull are evaluated via measuring the deflection of the springs. In the numerical model only two degrees of freedom are allowed, the others are completely fixed. No springs are modelled. Forces are evaluated via the pressure forces acting on the ship hull.

- In the numerical model we used a half model to safe computational effort. The model was cut along the longitudinal axis; a symmetry plane was defined along the cutting plane. This is likely to lead to different results than a full model. Especially in the region of the jets which evolve above the filling nozzles, the flow regimes are not the same: in two middle rows, the nozzles are sloped towards the middle, so that the jets collide in the middle.

- Where the numerical model has the scale of the prototype, the physical model was scaled according to the Froude similarity with a scale of 1: 25.

Viscous forces are assumed to be negligible. With that, scale effects can not completely be avoided.

7 CONCLUSION AND OUTLOOK

In this study the filling process of a ship lock was performed including a movable ship hull. For that a sequence of several mesh manipulation steps was used to pre-process the complex filling system simulation domain. The mesh was divided into two zones, a movable zone and a static zone, which are connected through two interfaces. The moving zone is a rectangular-shaped block, vertically able to slide along its interface to the static zone. It includes the ship hull boundaries and allows cell deformation due to motion of the hull as well as layer addition from the bottom. For the filling, a vertical plane in the longitudinal culvert was defined as valve, enabling the blockage or unblocking of fluxes through the faces.

The simulation of the filling process showed little agreement with the measurement data of a physical scale model of the lock. This can at least partly be blamed on the coarse mesh, used for the first results. Simulations with better mesh resolution are currently prepared. The possible reasons for the deviation between the physical and the numerical model results were listed. For better comparison, the differences between the models should be minimized. Future research should consider investigating the influence of the different degrees of freedom e.g. by modelling the springs in the numerical model. Scale effects could be avoided by modelling the same scale as the physical model or adjusting viscosity in the numerical model. Further, a complete model of the lock should be simulated, to avoid errors due to the symmetry plane and the influence of the turbulence modelling could be investigated by testing alternatives.

Simulation time and pre-processing effort make the procedure for simulating the filling of the lock with movable ship very complex and expensive. The setup further requires lots of expert knowledge. Since lots of geometry dependent adaptions in the mesh are necessary, the process cannot be simply automated and transferred to any example. For making the shown procedure more practical and reach broader applicability, the single steps have to be combined and generalized.

8 PUBLICATION BIBLIOGRAPHY

Beaudoin, Martin; Jasak, Hrvoje (2008): Development of a generalized grid interface for turbomachinery simulations with OpenFOAM. In : Open source CFD International conference, vol. 2.

InCom Working Group 155 (2015): Ship Behaviour in Locks and Lock Approaches. PIANC Report No. 155. With assistance of Carsten Thorenz, D. Bousmar, J.-P. Dubbelman, Li Jun, D. Spitzer, J. J. Veldman et al.: PIANC - The World Association for Waterborne Transport Infrastructure.

Jasak, Hrvoje; Tukovic, Zeljko (2010): Dynamic mesh handling in openfoam applied to fluid-structure interaction simulations. In J. C. F. Pereira, A. Sequeira (Eds.). V European Conference Computational Fluid Dynamics, ECOMAS CFD 2010. Lisbon, Portugal, 14-17 June, pp. 1–19. Available online at http://congress.cimne.com/eccomas/cfd2010/papers/01178.pdf, checked on 7/1/2015.

Kemnitz, Bernhard (2008): Gutachten über das Füll- und Entleersystem des Neubaus der Weserschleuse in Minden. Federal Waterways Engineering and Research Institute (Bundesanstalt für Wasserbau). Karlsruhe, Germany.

Partenscky, H.-W. (1986): Binnenverkehrswasserbau - Schleusenanlagen. Berlin, Heidelberg: Springer Verlag.

Rusche, Henrik (2002): Computational Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions. Doctoral Thesis. Imperial College of Science, Technology & Medicine, London, Great Britain. Department of Mechanical Engineering. Available online at http://powerlab.fsb.hr/ped/kturbo/OpenFOAM/docs/HenrikRuschePhD2002.pdf, checked on 1/6/2015.

Sieber, Galina (2002): Numerical simulation of fluid-structure interaction using loose coupling methods. Dissertation. TU Darmstadt, Darmstadt. Fachbereich Maschinenbau. Available online at http://tuprints.ulb.tu-darmstadt.de/254/1/thesis1.pdf.

Thorenz, Carsten; Strybny, Jann (2012): On the numerical modelling of filling-emptying system for locks. In R.-P. Hinkelmann, Y. Liong, D. Savic, M. H. Nasermoaddeli, K.-F. Daemrich, P. Fröhle, D. Jacob (Eds.): Proceedings of 10th International Conference on Hydroinformatics 2012. Hamburg, Germany. International Conference on Hydroinformatics. Hamburg: TuTech-Verl.