Bundesanstalt für Wasserbau (Hg.)

OpenFOAM® in Hydraulic Engineering - Book of Abstracts

BAW-Workshop, November 21 and 22, 2018 in Karlsruhe. Book of Abstracts

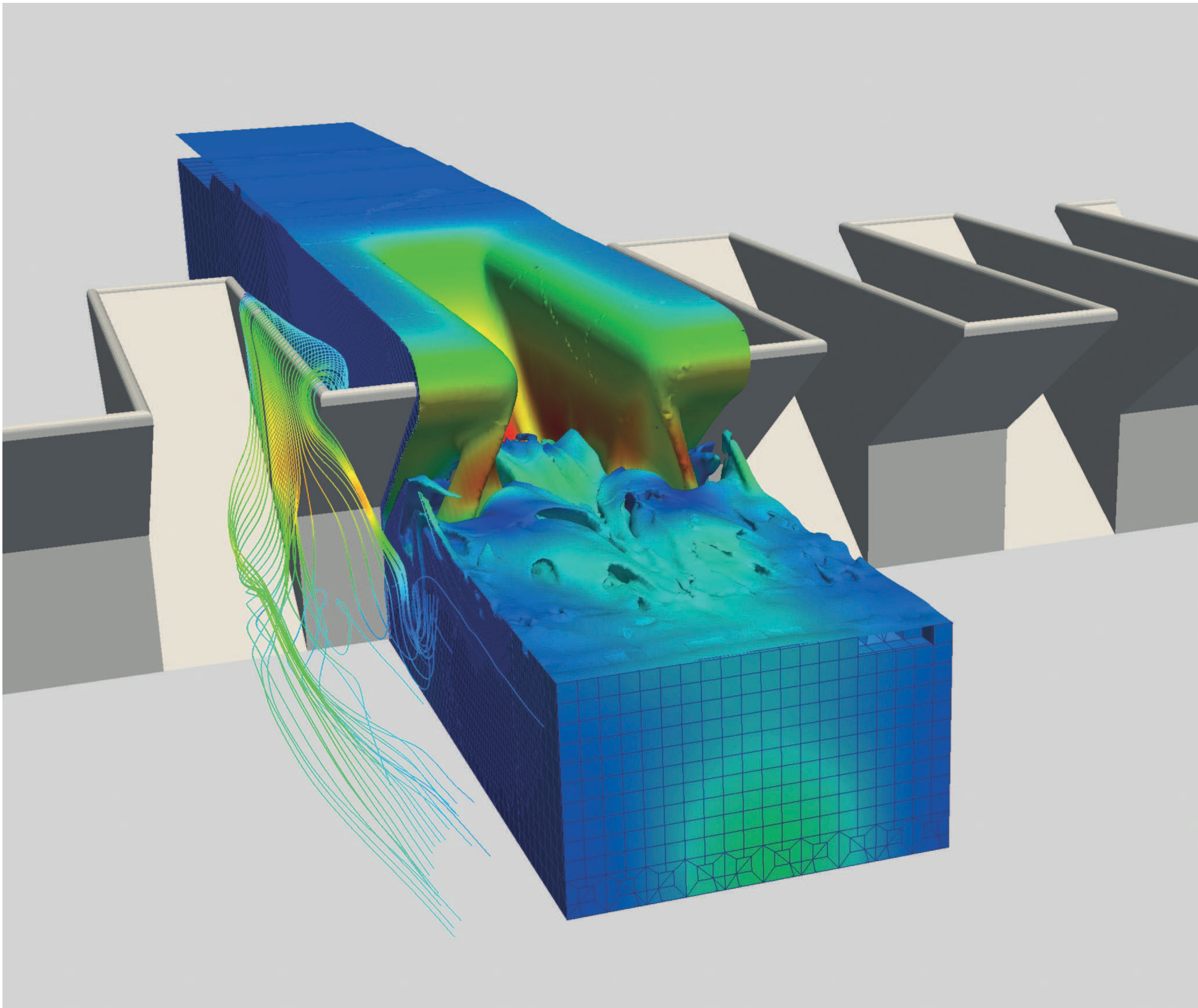
BAW
Federal Waterways Engineering
and Research Institute

BAWWorkshop

Book of Abstracts

OpenFOAM® in Hydraulic Engineering

November 21 and 22, 2018



BAWWorkshop

OpenFOAM® in Hydraulic Engineering

Programme

Wednesday, November 21, 2018

12:30	Registration and Coffee
13:00	Welcome Address *Lydia Schulze (BAW)*
13:15	Introduction Round
13:40	How to benefit from OpenFOAM® capabilities in custom software projects *David Gisen (BAW)*
14:05	The IsoAdvector VoF method: Recent and future development *Dr. Johan Rønby (Aalborg University)*
14:30	Coffee Break
15:00	Challenges of modelling a large-scale model with hydraulic structure and floodplain *Jakob Herbst (BAW)*
15:25	Immersed Boundary Surface Method foam-extend *Prof. Dr. Hrvoje Jasak (University of Zagreb)*
15:50	Integration of moving objects into the 3D modelling of lockage processes *Torsten Hartung (BAW)*
16:15	Coffee Break
16:45	Modelling of attraction flow for upstream fish passage *Markus Zinkhahn (BAW)*
17:10	Numerical modelling of air entrainment in stepped spillways *Silje Kreken Almeland (Norwegian University of Science and Technology)*
17:35	3D CFD simulations for the spillway design for the dam Friedrichswalde-Ottendorf *Max Heß (Technical University Nürnberg)*
18:05	Overview of the computing and visualisation infrastructure of the BAW *Fabian Belzner (BAW)*
19:00	Dinner

BAW
Federal Waterways Engineering
and Research Institute

Competence for Waterways

BAWWorkshop
OpenFOAM® in Hydraulic Engineering

Programme

Thursday, November 22, 2018

08:30	Welcome Address *Dr.-Ing. Carsten Thorenz (BAW)*
08:35	Aeration of free falling flows – Development of an air entrainment model *Markus Wagner (BAW, KIT)*
08:45	Application of overset mesh for simulating fluid-structure interaction using foam-extend 4.1 *Dr. Željko Tuković (University of Zagreb)*
09:10	Simulation of flow-induced vibrations of a radial gate with underflow *Georg Göbel (BAW)*
09:35	Coffee Break
10:05	Next generation of waves2Foam *Dr. Niels Jacobsen (Deltares)*
10:30	Ship propulsors: accuracy vs. speeds *Tarek Beck (BAW)*
10:55	Coffee Break
11:25	Modelling density induced ship forces during lockage *Dr.-Ing. Carsten Thorenz (BAW)*
11:50	Examination of critical erosion velocities in spherical sediment with OpenFOAM® *Arne Daniel (University Emden/Leer)*
12:15	Closing *Lydia Schulze (BAW)*
12:30	Optional: Lunch (self-payer)
13:30	Optional: Tour through BAW labs

Preface

The OpenFOAM® library provides an excellent toolbox for a large variety of CFD applications. In the last years Open-FOAM® gained influence in the field of hydraulic engineering. At the Federal Waterways Engineering and Research Institute (BAW) OpenFOAM® is used for daily project work and in several research projects. The application field is very wide: it ranges from the simulation of complex flows in the near-field of hydraulic structures over the simulation of the fluid-structure interaction at vibrating dams to the investigation of the outflow of ship turbines.

With our workshop OpenFOAM® in Hydraulic Engineering at the BAW in Karlsruhe we want to create a platform for knowledge transfer for users and researches working with OpenFOAM® in the field of hydraulic engineering. The workshop aims to provide an opportunity to present the latest research and/or give insights into daily project work or special application cases with the OpenFOAM® toolbox.

This booklet collects abstracts of the talks presented at the BAW Workshop held at the Federal Waterways Engineering and Research Institute in Karlsruhe, November 21/ 22.

How to benefit from OpenFOAM capabilities in custom software projects

Author: David Gisen, Bundesanstalt für Wasserbau

OpenFOAM consists of numerous libraries written in C++, providing a toolbox for all kinds of numerical calculation. For engineers with little training in programming, it can be difficult to find and apply these tools because the authors distributed the code in small chunks for reusability (Jasak et al. 2007). I aim to lower this hurdle by describing the approach I use. For my PhD thesis, I built an individual-based model of fish moving in a laboratory flume (Gisen 2018). Model fish took input stored on an OpenFOAM mesh, e.g. velocity, which had to be interpolated to the fish center. But how?

I found the most convenient ways of searching the code to be the OpenFOAM C++ Source Code Guide (https://cpp.openfoam.org, starting with 3.0.1) and the GitHub repository (https://github.com/OpenFOAM?tab=repositories, starting with 2.0.x). The guide is recommended for basic research, as its results are clearer and it contains descriptions of member functions. The repository is more suitable for digging through classes located in neighbour directories, as its navigation layout is clearer.

The first step to find suitable functions is to search for keywords similar to the desired function name in the guide. For all results, member function text descriptions have to be evaluated. If they match the task, they can be inserted in custom C++ code. Following inclusion of the headers, the code has to be compiled and linked. This can be done by using OpenFOAM's wmake tool with an options file or a custom makefile. Anyway, they have to contain the paths of the header files (.H) and compiled libraries (.so) of every function library included. The basic path is known from the guide. Then, the corresponding folder lnInclude has to be added to the EXE_INC variable. The same goes for the respective libraries at $FOAM_LIBBIN, which are added to EXE_LIBS. After successful compilation and linkage, the new executable can be applied to produce data, for example model fish tracks (Figure 1).

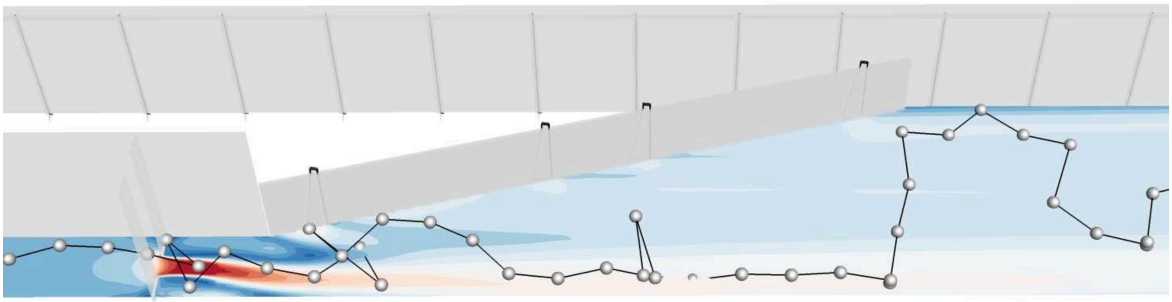


Figure 1: Model fish track in a hydraulic flume. Flow from left to right, fish movement vice versa.

Literature

Gisen, D.C. (2018): Modeling upstream fish migration in small-scale using the Eulerian-Lagrangian-agent method (ELAM). Dissertation Universität der Bundeswehr München, in press, https://hdl.handle.net/20.500.11970/105158.

Jasak, H.; Jemcov, A.; Tuković, Ž. (2007). OpenFOAM: A C++ library for complex physics simulations. In: Proc. International Workshop on Coupled Methods in Numerical Dynamics. IUC, Dubrovnik, Croatia.

The IsoAdvector VoF method: Recent and future development

Author: Johan Roenby, Department of Mathematical Sciences, Aalborg University

IsoAdvector is a geometric Volume-of-Fluid (VoF) method developed as an alternative to the MULES limited interface compression method used for interface advection in the interFoam family of multiphase flows solvers. The isoAdvector method was first published in Roenby et al. (2016), where it was shown to be superior in terms of accuracy, interface sharpness, and even in terms of efficiency, for a number of standard interface advection test cases. As of OpenFOAM-v1706 the method was integrated into the official OpenFOAM source code, where it now forms the basis for the interIsoFoam solver. Since its first appearance, isoAdvector has been tested and applied in several different engineering contexts, see e.g. Laurila et al. (in press), Roenby et al. (2016), Meredith et al. (2017), Gatin et al. (2018) and Letout et al. (2017). Based on the experience gained from these tests, the method has been improved and undergone further development (e.g. Scheufler and J. Roenby (2018), Vukčević (2018)). Most recently in OpenFOAM-v1806 it has been extended to work with adaptive mesh refinement (AMR).

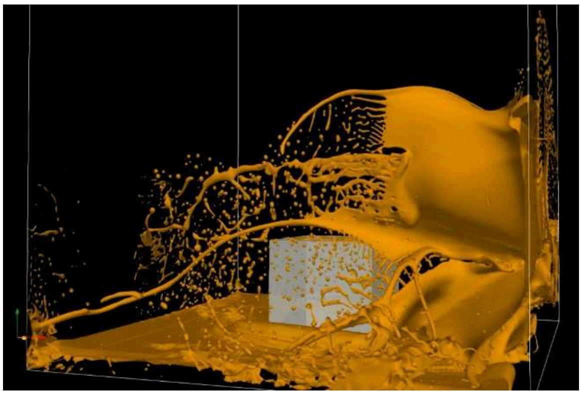


Figure 1: Snapshot from a refined version of the damBreakWithObstacle tutorial using interIsoFoam with AMR.

In this presentation, I will give a brief overview of the isoAdvector method and its applicability on general structured and unstructured meshes. I will then show examples of recent practical applications and recent improvements to the method. Finally, I will discuss directions for future research aimed at improving interfacial flow simulations in OpenFOAM.

Literature

Gatin, I.; Vukčević, V.; Jasak, H.; Seo, J.; Rhee, S. H. (2018): CFD verification and validation of green sea loads, Ocean Engineering, vol. 148, pp. 500–515.

Laurila, E.; Roenby, J.; Maakala, V.; Vuorinen V. (in press): Large-Eddy Simulation of a Pressure-Swirl Atomizer, International Journal of Multiphase Flow.

Letout, S.; Ratvik, A.-P.; Tangstad, M.; Johansen, S.-T.; Olsen, J.-E. (2017): A Multiscale Numerical Approach of the Dripping Slag in the Coke Bed Zone of a Pilot Scale Si-Mn Furnace, in Progress in Applied CFD – CFD2017.

Meredith, K. V.; Zhou, X.; Wang, Y. (2017): Towards Resolving the Atomization Process of an Idealized Fire Sprinkler with VOF Modeling, in ILASS2017 - 28th European Conference on Liquid Atomization and Spray Systems.

Roenby, J.; Bredmose, H.; Jasak, H. (2016): A computational method for sharp interface advection, Royal Society Open Science, vol. 3, no. 11, p. 160405.

Roenby, J.; Larsen, B. E.; Bredmose, H.; Jasak H. (2017): A new Volume-of-Fluid method in OpenFOAM, in VII International Conference on Computational Methods in Marine Engineering.

Scheufler, H.; Roenby, J. (2018): Accurate and efficient surface reconstruction from volume fraction data on general meshes, arXiv:1801.05382 [physics].

Vukčević, V.; Roenby, J.; Gatin, I.; Jasak H. (2018): A Sharp Free Surface Finite Volume Method Applied to Gravity Wave Flows, arXiv:1804.01130 [physics].

Challenges of modelling a large-scale model with hydraulic structure and floodplain

Demonstrated at the example of the barrage Nassau at the river Lahn

Author: Jakob Herbst, Bundesanstalt für Wasserbau

In the presented study a 3D-numerical simulation on a large-scale river section for the barrage Nassau at the river Lahn was performed with OpenFOAM. The consisting structure has reached the end of useful life and will be replaced by an inflatable rubber dam. The aim of the numerical simulation was to estimate the influence of the new structure on the upstream water level. Additionally the changing flow conditions in the entrance corridor of the lock were focused.

From the prototype to the numerical model different tasks needed to be managed. On the basis of the provided data such as a topographic scan of the area, construction plans and hydrological datasets, a computational domain and a grid were defined. The 1 km long river section was calibrated by varying boundary conditions on the model inflow and outflow, on the river bed and on the floodplain. For the evaluation of the results sophisticated post-processing procedures were used. The applicability of the method and remaining questions will be discussed.

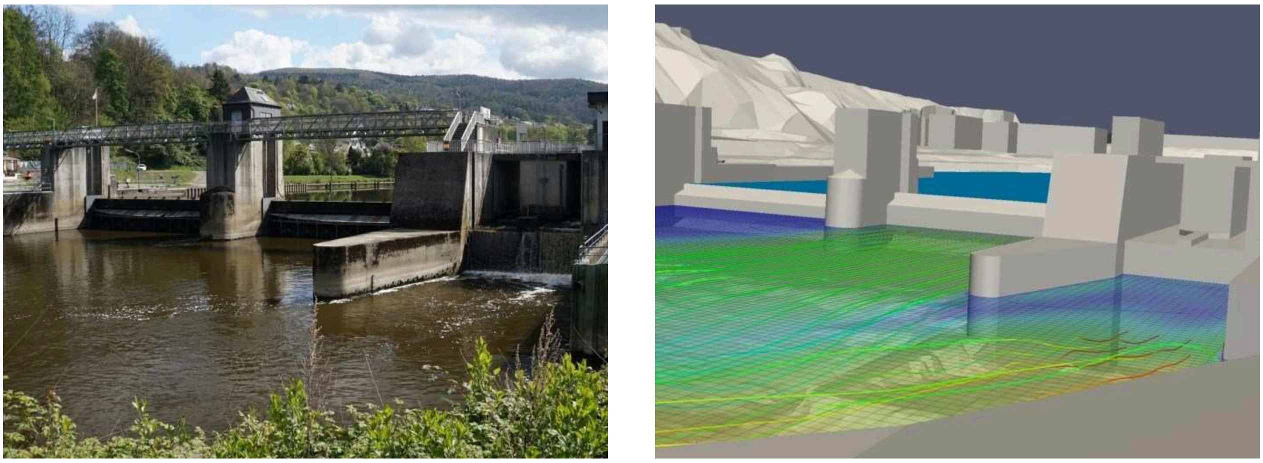


Figure 1: The prototype of the barrage Nassau at the river Lahn (left) and the numerical model of the structure (right).

Immersed Boundary Surface Method foam-extend

Author: Prof. Hrvoje Jasak, University of Zagreb, Croatia

Dynamic mesh problems in modern Computational Fluid Dynamics (CFD) today are becoming increasingly complicated, both in terms of boundary-to-flow interaction and complex physics support. The adage of a choice between complex geometry with simple flow physics or simple geometry for complex physics is less and less acceptable in engineering practice. While the conventional methods of dynamic mesh deformation and preserved connectivity or topological mesh changes seem versatile and robust, they suffer from geometrical limitations in the freedom of boundary motion and boundary-to-boundary interaction. Further, traditional dynamic mesh setup demands significant user interaction and may fail without warning, due to limitations of user-defined motion / topological changes or unforeseen dynamic mesh states. Particularly problematic are cases of arbitrary solution-dependent motion and deformation, bodies in close vicinity and in particular the problems of boundary-to-boundary collision, which cannot be adequately treated within this framework.

Immersed Boundary Method (IBM) is a possible alternative to moving deforming meshes in CFD. It accounts for presence of boundaries within a computational domain without a body-fitted mesh structure. Instead, modifications in the discretised equations are performed at matrix level to account for presence of "internal boundaries". Methods vary in the manner in which the presence of the boundary in the volume mesh is accounted for, as well as the spatial description of the discontinuous solution fields, either in the volume or at the boundary. Examples include continuous or discrete forcing methods, with indirect and direct imposition of boundary conditions (BC-s).

While conventional IBM has been in use for decades, close inspection shows significant drawbacks in each of its many variants, leaving scope for further improvement. Typically, they relate to the ability of the method to evaluate surface forces, preserve boundedness of bounded scalars, guarantee no flux through impermeable walls, support conventional turbulence modelling techniques such as wall damping and the manner of resolving non-polynomial solution functions, such as the bounded step-change in the Volume of Fluid (VoF) variable. The most promising of the IBM methods is the direct forcing approach with direct imposition of boundary conditions, which was previously implemented by the author. However, current experience with complex models and non-trivial physics shows it is still insufficiently robust for practical engineering use.

Based on the existing experience of direct-forcing direct-BC IBM, a new Immersed Boundary Surface (IBS) method is developed, implemented and validated in foam-extend-4.1 and will be the topic of this talk. The method combines the properties of body-fitted polyhedral background mesh and Immersed Boundary (IB) without computational overhead or geometrical simplification.

The IBS method avoids the problems of boundary fitting or equation forcing in other IBM techniques by mimicking the presence of additional immersed boundary faces in the mesh at the matrix level without the need for detailed geometrical description. It is stable and conservative, with compact computational stencil support and strict boundedness of variables. Possible geometrical problems in precise surface-to-cell intersections are handled by robust error-avoidance algorithms, capable of detecting and resolving common cutting errors such as direct point-on-point or edge-on-edge hit.

The basis of the algorithm is the adjustment of mesh metrics (cell volumes, face areas) and geometry (cell and face centres, weighting factors and delta coefficients) for the cells interacting with the IB surface, without the need to modify the mesh connectivity. Deactivated cells are treated within the same framework without the need for special practice. Boundary conditions in the IB surface are imposed in a “body-fitted” manner without simplification. Since the algorithm exactly mimics the presence of body-fitted faces next to the immersed boundary, conventional near-wall treatment such as law-of-the-wall or other forms of wall damping can be used without modification.

The IBS method is extended to moving deforming immersed boundary cases, which handle the space conservation law with robust error-avoidance equivalent to the static surface cutting cases. This is validated on moving deforming IBS cases which do not exhibit the Courant-Friedrichs-Levy (CFL) limitation in boundary motion.

The IBS algorithm operates in parallel without algorithmic degradation, since it is inherently local. It is also equipped with dynamic mesh refinement and coarsening for polyhedral cells in 2-D and 3-D, with optional load balancing.

Validation cases presented in this study range from simple Laplace and potential flow equation, laminar and turbulent flow in 2-D and 3-D. The focus of the study is the simulation of floating objects in free surface flow, where it is necessary to show the ability of the algorithm to preserve mass and boundedness of scalars, accurately evaluate forces on the IB surface and couple the precise evaluation of near-wall pressure and velocity with coupling to 6-Degree-of-Freedom (6-DoF) floating body simulations, which also serve as the ultimate test case.

Integration of moving objects into the 3D modelling of lockage processes

Author: Torsten Hartung, Bundesanstalt für Wasserbau

Locks are key elements to overcome local differences in the water level at barrage controlled waterways or canals. In particular, the lifting heights at the federal waterways differ from a few meters at the naval locks of the Kiel-Canal up to 38 meters at a proposed lock in Lüneburg. The requirements for the hydraulic system are versatile. Fast lock cycle times and the safety of passing ships on the one hand have to be consistent with reasonable costs for construction and operation on the other hand. Meeting all these requirements is part of the optimisation process.

The forces acting on the ship during lockage is a critical design criterion. For their determination the Federal Waterways Engineering and Research Institute (BAW) uses different methods including physical and numerical models. Despite continuous development of numerical methods, the modelling of the filling process in a navigation lock considering a floating ship and opening valves is still a challenging task. Due to the large vertical movement of the lifted ship and the small spaces between the ship hull and the lock chamber walls, special modelling concepts become necessary. Each of these concepts has its own advantages and disadvantages.

A common approach and a good starting point to handle moving objects throughout the lockage is the deforming-mesh method. Vertices of the mesh are displaced during the simulation according to the motion of the ship's boundaries. However, the lifting height is limited to a few meters with this approach due to the resulting mesh distortion which leads to a collapsing mesh. More sophisticated methods like overset-mesh or immersed-boundary are needed to overcome this major drawback. Finally, the increasing complexity of the numerical methods must not affect the main outcome which are the forces acting on the ship during lockage.

Different methods are currently investigated at the BAW with respect to their suitability for the simulation of lockage processes. The outcome of all these methods has to be consistent with each other as well as validated. For that the BAW uses physical scale models with integrated measurement structures to determine the acting forces on the ship. After all it is the ultimate goal to have the choice of considering lockage processes with the means of physical models in the laboratory or numerical models.

Modelling of attraction flow for upstream fish passage

Author: Markus Zinkhahn, Bundesanstalt für Wasserbau

To restore river connectivity, functional fishways at hydropower dams are indispensable. Functionality strongly depends on the ability of migrating fish to find the entrance of the fishway without major delays. In order to increase attraction efficiency in competition to the bulk discharge from the draft tubes, auxiliary water is supplied into the entrance pool and thus an attraction flow perceptible to the fish is generated. Its propagation in the tailrace is influenced by the mostly heterogeneous and strongly secondary flow dominated velocity field coming from the draft tubes. The challenge is to determine the minimum required flow rate to reach a certain distance with the attraction flow.

To answer this question we use numerical models created with the interFoam solver from the OpenFOAM library. To achieve a high prognostic ability, the models must be calibrated for the relevant discharge conditions using suitable velocity measurements. Calibration is carried out by varying the velocity field at the inlet boundary of the draft tube. In addition to the swirl boundary condition used in Gisen (2017) (Fig. 1; 5c), simplified (Fig. 1; 5a and 5b) and more complex approaches (Fig. 1; 5d) for the velocity field at the boundary are tested. In conclusion the suitability of a boundary type depends on the flow conditions at site, the available data and the desired accuracy.

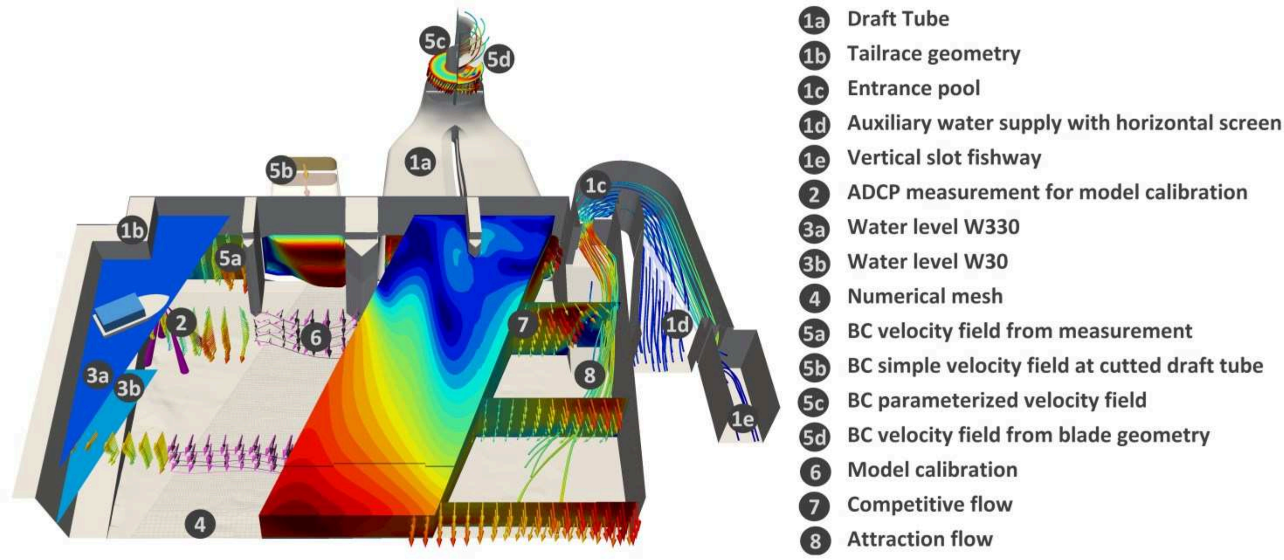


Figure 1: *Overview of main components of a hydropower tailrace model and possible boundary conditions for the draft tube flow to reproduce the competitive flow adequately.*

Literature

Gisen, D.; Weichert, R.; Nestler, J. (2017): Optimizing attraction flow for upstream fish passage at a hydropower dam employing 3D Detached-Eddy Simulation. In: *Ecological Engineering* 100, p. 344–353. https://hdl.handle.net/20.500.11970/104574.

Numerical modelling of air entrainment in stepped spillways

Author: Silje Kreken Almeland, Norwegian University of Science and Technology

The amount of air entrained in the flow along a hydraulic structure constitutes an important design criterion for hydraulic structures like a stepped spillway. Air entrained in the water flow will change the properties of the flow and will influence the performance of the hydraulic structure. It might be an advantage to be able to predict the amount of air entrained in the flow, both to optimise the structure regarding safety and performance. Traditionally, numerical modelling of free-surface air-water flows have been treated by interface capturing methods, like level set and volume of fluid (VoF). Nevertheless, when the turbulence is high enough in free-surface flow, air will be entrained in the water phase and transported with the water flow. The scales involved in the air entrainment process are in the range of millimetres. This means that a very fine grid is needed to capture these processes. When the grid is not fine enough, the processes at the surface will not be captured, and the amount of air within the water phase will be underestimated. One research path to model air entrainment based on interface capturing methods is to introduce a sub grid model to introduce air into the water phase at relevant locations. An early attempt following this road was done by Hirt (2003). In this air entrainment model, air is entrained in the flow based on whether the stabilising forces of gravity and surface force are overcome by the perturbing forces of turbulence. This model was reported to correctly identify when air is to be incorporated to the flow for a stepped spillway (Valero & Bung, 2015). Another attempt in this direction was done by Lopes, Leandro, and Carvalho (2017). In this model an extra advection equation for the dispersed air phase was added to the algorithm. In this equation, a source term was added to account for the air entrainment. In the current work the air entrainment model by Lopes et al. (2017) is updated to OpenFOAM version 5.0, and applied to a stepped spillway. The source term developed by Hirt (2003) is added to the model as a user choice. Results of applying the different source term frameworks are compared to each other, and to experimental data. The results show that the air entrainment models are able to reproduce the inception point and surface elevation reasonably good.

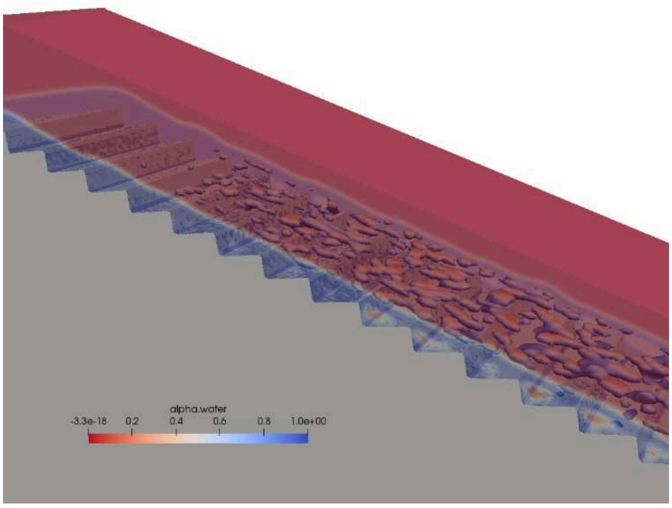


Figure 1: Air entrainment in a stepped spillway.

Literature

Hirt, C. (2003). Modeling turbulent entrainment of air at a free surface. Flow Science, Inc.

Lopes, P.; Leandro, J.; Carvalho, R. F. (2017, dec). Self-aeration modelling using a sub-grid volume-of-fluid model. International Journal of Nonlinear Sciences and Numerical Simulation, 18(7-8). Retrieved from https://doi.org/10.1515/ijnsns-2017-0015

Valero, D.; Bung, D. (2015). Hybrid Investigation of Air Transport Processes in Moderately Sloped Stepped Spillway. E-proceedings of the 36th IAHR World Congress 28 June – 3 July, 2015, The Hague, the Netherlands.

3d CFD simulations for the spillway design for the dam Friedrichswalde-Ottendorf

Author: Max Heß, Technische Hochschule Nürnberg, Institut für Wasserbau und Wasserwirtschaft

Updated statistics based on recent flood events increase the design discharges for spillways and cause the need to recover the flood safety for those structures. In order to regain the hydraulic effectiveness of the flood retention reservoir Friedrichswalde-Ottendorf in the Free State of Saxony (Germany) investigations on three-dimensional hydrodynamic-numerical simulations have been performed. With the usage of the open source CFD-code OpenFOAM®, executed at the cluster systems of the Erlangen Regional Computing Center (RRZE) and of the Institute for Hydraulic Engineering and Water Resources Management of the Technische Hochschule Nürnberg (IWWN), numerical models have been developed to be investigated in terms of hydraulic functionality of structural variations.

Prior to the numerical simulations several investigations on a physical model (longitudinal scale 1:25) were conducted at the hydraulic laboratory of the IWWN. The geometry of the spillway could be captured from the physical model by the usage of a 3d-laser scanner. The gained data represented by a point cloud of about 7 Mio. points had to be translated to a surface-based CAD-file to allow the compatibility with OpenFOAM based mesh generation. During the mesh generation process the surface geometry of the spillway, scaled to a ratio of 1:1, was included using the meshing tool snappyHexMesh. With a total number of approx. 6 Mio. control volumes the discretized numerical model includes different sizes of cells as well as layer cells at no-slip walls.

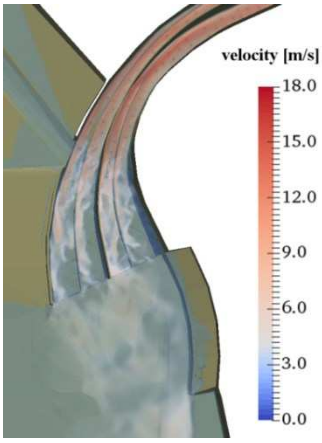


Figure 1: Stilling basin of the spillway Friedrichswalde-Ottendorf with structural elements.

For the numerical investigations the computational domain needs to include the two fluid phases water and air. Therefore the solver interFoam was used which runs with a volume of fluid method to detect the free surface. For the flow entering the domain a flow rate boundary condition (BC) were set. To provide a certain water level at the outlet an existing OpenFOAM BC had been modified so that the desired elevation of the free surface could be set by a new defined variable. Based on the highly turbulent flow conditions in stilling basins the simulations needed to be performed under the usage of a turbulence modelling method. Regarding this, the simulations were executed i. a. with a detached eddy simulation (DES) approach so that the larger eddies are calculated directly and only the smaller eddies are modeled in the free stream areas.

Aeration of free falling flows
Development of an air entrainment model

Author: Markus Wagner, Bundesanstalt für Wasserbau, Karlsruhe Institute for Technology

The evaluation of downstream water levels for a safe downstream fish passage is based on a rule of thumb, derived from water turbine fish passage regulations. Thorenz et al. (2018) performed numerical simulations to evaluate these rules and concluded that they are not sufficient to guarantee safe fish passage over weirs. Therefore, there is a need for numerical simulations to evaluate the hydraulic situation of fish passage situations.

The currently available two-phase flow solvers which are based on the volume of fluid method (VOF) fail at big scale self-aerated flows such as shown in Figure 1. This is due to the multi scale flow region with largely stretched water surface regions and small air bubbles. Air entrainment prediction would need a very fine discretisation of the water-air interface. This would lead to a huge computational effort. However, the correct prediction of the four air transport processes, as shown in Figure 1, is essential for the hydraulic situation inside the water jet and for the further course of the downstream flow. To predict surface aeration (1) further development of the VOF-solver interFoam is necessary. This is part of the current research project. Local aeration (2) can be realized by a proper discretisation of the immersion point. A model for air transport (3) and detrainment (4) was already developed by Schulze (2018). The reasonable representation of air entraining, transport and detraining processes give the opportunity to evaluate overflown weirs regarding to the immersion depth of the water jet and turbulent flow conditions downstream.

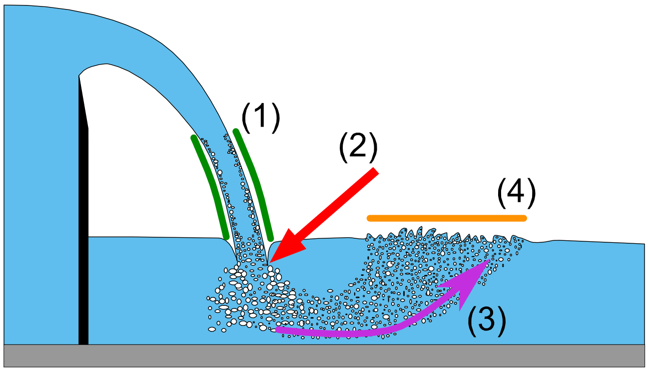


Figure 1: Air transport of a plunging weir overflow. Surface aeration (1), local aeration (2), air transport (3) and air detrainment (4).

Literature

Thorenz, C.; Gebhardt, M.; Weichert, R. (2018): Numerical Study on the Hydraulic Conditions for Species Migrating Downstream over a Weir.

Schulze, L. (2018): Development of an Application-Oriented Approach for Two-Phase Modelling in Hydraulic Engineering, Dresdner Wasserbauliche Mitteilungen, (61).

Application of overset mesh for simulating fluid-structure interaction using foam-extend 4.1

Author: Željko Tuković, University of Zagreb, Croatia

It is presented an extension of the open-source fluid-structure interaction (FSI) toolbox **solids4foam** to enable application of the overset mesh methodology recently implemented in the foam-extend. In the solids4foam toolbox, the fluid flow is described by the Navier-Stokes equations in the arbitrary Lagrangian-Eulerian form, while the momentum equation in the total or updated Lagrangian form is used to describe the solid deformation. Both the fluid and the solid are discretised in space using the second-order accurate cell-centred finite volume method and the second-order accurate implicit scheme is used for temporal discretization. The adjustment of the fluid mesh to the time varying shape of the deformable body is usually carried out by mesh deformation, where the internal mesh vertices are moved based on the prescribed motion of the boundary vertices, while the topology of the mesh stays unchanged. In the overset mesh approach, the fluid mesh is composed of one background mesh and one or more foreground meshes attached to the moving/deforming bodies. Motion of the foreground meshes independently from the background mesh should enable much higher motion amplitudes during FSI simulations. Application of overset mesh in the FSI calculation framework required series adjustments related to the mesh deformation solvers and support for high order temporal discretization schemes.

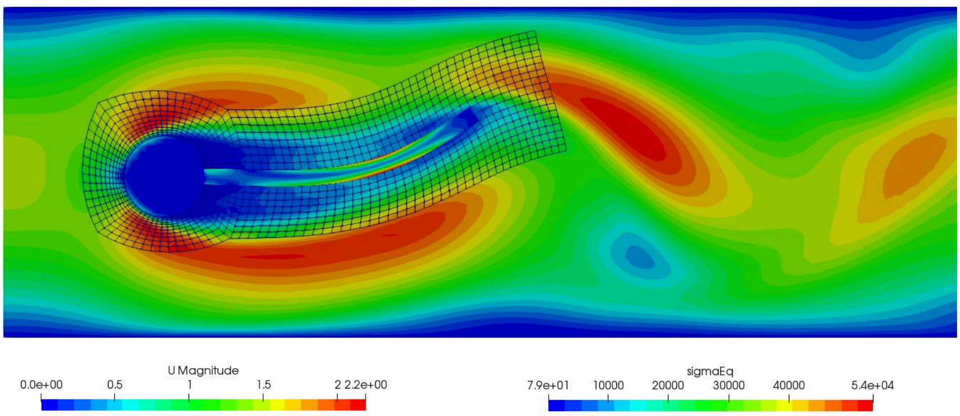


Figure 1: Turek and Hron FSI-2 test case: foreground mesh deformed by RBF mesh motion solver.

Literature

Cardiff, P.; Karač, A.; De Jaeger, P; Jasak, H.; Nagy, J.; Ivanković, A; Tuković, Ž (2018): An open-source finite volume toolbox for solid mechanics and fluid-solid interaction simulations. In: Computer Physics Communications, submitted for review. Available online at https://arxiv.org/abs/1808.10736v2.

Tuković, Ž.; Jasak, H.; Karač, A.; Cardiff, P.; Ivanković, A. (2018): OpenFOAM finite volume solver for fluid-solid interaction. In: Transactions of Famena, 42 (3), pp. 1-31. Available online at https://hrcak.srce.hr/206941.

Simulation of flow-induced vibrations of a radial gate with underflow

Author: Georg Göbel, Bundesanstalt für Wasserbau

In hydraulic engineering, different sources of flow-induced vibrations exist. Vibrations of a hydraulic structure can be caused by turbulences in the approach flow, by instabilities arising at the structure itself or by the coupling of the surrounding flow with the movement of the structure. While the exciting force in the flow exists even if the structure does not vibrate in the first two cases, the third one is more complicated: the forces in the flow arise due to the movement of the structure and the vibration of the structure is driven by the forces in the flow. Such loop of mutual control is referred as self-excitation or in the field of hydraulic engineering sometimes called movement-induced excitation. (Naudascher and Rockwell 1994)

The presented project deals with self-excited vibrations on a radial gate with underflow. The gate is known to vibrate under certain conditions with small gap width and low tailwater levels. For the numerical modelling of the gate, the interDyMFoam solver was used. This solver is part of the widely used open-source CFD software OpenFOAM and provides an efficient tool for coupling solid body motion with free-surface flow. Since the vibration of the modelled gate was a rotational vibration around the trunnion points, the model could be reduced to two dimensions. A fixed water level was used as boundary condition on the inflow and outflow boundary. Even though the full coupling is inevitable for self-excited vibrations, the coupled simulation is restricted in some points due to the complex nature of the prerequisite conditions. The small openings that are typically prone to self-excited vibrations require locally a very fine mesh. Additionally, high pressure gradients occur both in time and space due to the structural movement. These factors both have a negative effect on the stability of the simulation. Therefore, the free vibration of the gate is restricted to a prescribed vibration with a fixed frequency and amplitude. This improves the stability of the simulation but additional data about the predominant vibration is needed. The application of a 2D model brings another restriction to the simulation; the use of a LES-type turbulence model is not feasible therefore a k-ω-model is applied. The highly turbulent flow regime beneath the gate is thereby simplified and small scale vortices that occur in nature cannot be represented in the simulation.

Data from an extensive measuring campaign was used to calibrate the numerical model. Despite the restrictions in the model, the results look reasonable related to the pressure fluctuation beneath the gate. The described method serves as a useful engineering tool to check gate design for self-excited vibrations. Furthermore, the results enable an interesting view on the underlying process of interaction between vibrating gate and flow.

Literature

Naudascher, E.; Rockwell, D. (1994): Flow-induced vibrations. An engineering guide. Rotterdam: A.A. Balkema (IAHR series of Hydraulic Structures Design Manuals, Volume 7).

Next generation of waves2Foam

Author: Niels G. Jacobsen, Deltares, The Netherlands

The modelling of free surface flows for riverine, coastal or offshore applications is often faced with the challenge of absorbing any internal disturbance due to initial conditions or outgoing waves at the boundaries. This led to developments of numerical beaches, sponge layers, and absorbing boundary conditions (like perfectly matching layers and Sommerfeld-type conditions). In the framework of OpenFoam, the relaxation zone technique in waves2Foam (Jacobsen et al, 2012) and the shallow-water based boundary absorption in IHFoam/OlaFlow (Higuera et al., 2013) are widely used techniques. Neither method is perfect.

The waves2Foam framework allows for low reflection coefficients for a wide range of non-dimensional wave numbers (kh), but it comes at the cost of a considerable extension of the computational domain. Furthermore, it has proved difficult to apply the relaxation zone technique to absorb waves (disturbances) on an upstream boundary of a hydraulic structure, while achieving a fixed discharge without restraints to the water level. IHFoam/OlaFlow is based on the assumption of shallow-water waves, so the condition quickly results is unsatisfactory reflection coefficients.

In this work, the first steps of eliminating the deficiencies in the two existing methods are presented. The new development is a boundary-based absorption and generation method for 2DV applications, which can be tuned for reflection less than 5% over a wide range of kh, e.g. 0.0-2.5 (Borsboom and Jacobsen, In preparation). The starting point is the Sommerfeld transmissive boundary condition. In this context, 2DV means that the boundary condition is strictly derived for normal incident waves, though it is still possible to apply the absorption technique in 3D.

Applications of the method are presented, where the performance as an absorbing boundary condition is shown for regular and irregular waves. Furthermore, it is shown that the method allows for combined generation and absorption at the same boundary. Finally, results for the validation of flow over a hydraulically rough weir is shown, where the capability of absorbing translatory waves at the upstream boundary, while ensuring a target discharge without restraints on the upstream water level is shown.

Literature

Borsboom, M.J.A.; Jacobsen, N.G. (In preparation): A non-reflective boundary for normal incident, dispersive waves.

Higuera, P.; Lara, J.L.; Losada, I.J. (2013): Realistic wave generation and active wave absorption for Navier-Stokes models. Application of OpenFoam®. Coastal Engineering, 71, 102-118.

Jacobsen, N.G.; Fuhrman, D.R.; Fredsøe, J. (2012): A Wave Generation Toolbox for the Open-Source CFD Library: OpenFoam®. International Journal for Numerical Methods in Fluids, 70(9), 1073-1088.

Ship propulsors: accuracy vs. speed

Author: Tarek Beck, Bundesanstalt für Wasserbau

Bed and bank protection is part of the maintenance measures carried out on German inland waterways. Due to the fact that larger ships with stronger engines and propulsion systems are being developed the bed loads as well as the bed erosion will increase. Besides the wave loads and the near field velocities induced by the ship speed the propeller jet is one of the main reasons for bed or bank erosion, especially in manoeuvring situations. The uncertainties of experimental results, caused by the complex velocity field of the propeller jet and the grain size scaling effects, are difficult to determine. For this reason the Federal Waterways Engineering and Research Institute started a research project to investigate the river bed and bank erosion induced by the propeller jet applying numerical simulations. The aim of this project is to get a fully coupled CFD-DEM simulation including a moving ship, a rotating ducted propeller, realized by using a rotating object with the sliding interface method, and discrete particles to represent the bed and bank structures.

In order to simulate all the aspects induced by the combination of the ship hull, the ducted propeller, the rudder, the stream spread and the loads on the river bed and bank a complex numerical setup with a large and fine grid is needed. Furthermore, due to the complexity of the simulations a long simulation time will be required to estimate influences like the scour process on river beds and banks. One of the reasons for the long time needed is the gap between the ducted propeller and the nozzle which leads to small cells in combination with high velocities at the blade tips. The first improvement is to optimize the spatial resolution of the propeller and the nozzle. The second improvement is to modify the arrangement of the sliding interface between the rotor and stator grids to gain more space for larger cells. Finally the CFL number, different time schemes and iteration settings are investigated to receive the best fit between speed and accuracy. A Reynolds-Averaged-Navier-Stokes model with the OpenFOAM version 2.4.0 is used. The simulations are performed with three different propeller revolutions for five advance numbers. More than 400 different configurations are simulated to get a detailed overview of the uncertainties. Experimental results in open water conditions are used for the comparison.

Literature

Dong, X.Q.; Li, W.; Yang, C. J.; Noblesse, F. (2017): RANSE-based Simulation and Analysis of Scale Effects on Open-Water Performance of the PPTC-II Benchmark Propeller. In: Fifth International Symposium on Marine Propulsors.

Fangliang, W.; Yaling, P.; Zhang, Z.; Guodong, W. (2012): Application of Dynamic Mesh in Analysis of Propeller Hydrodynamic Characteristics. In: Applied Mechanics and Materials.

Modelling density induced ship forces during lockage

Author: Carsten Thorenz, Bundesanstalt für Wasserbau

For navigation locks, the forces on moored ships during the lockage are a most relevant design criterion, as they are a measure for the safety of the ships during the locking process. This has to be evaluated a-priori with physical or numerical models (e.g. Thorenz & Anke, 2013). For locks at the boundary between fresh and saltwater regions, the situation differs significantly from locks with a constant water density. For the new lock of Brunsbüttel at the Kiel-Canal, a major sea lock with a chamber length of 360 m and a design ship size of ~40000 t, physical mode tests were performed. The lock model was built with a scale of 1:47 and was operated with fresh water in order to test the lock performance. Later these tests were complemented by further numerical tests, which additionally included the impact of salinity. The numerical tests were performed on the basis of the software package OpenFOAM® 2.3.1 (Figure 1).

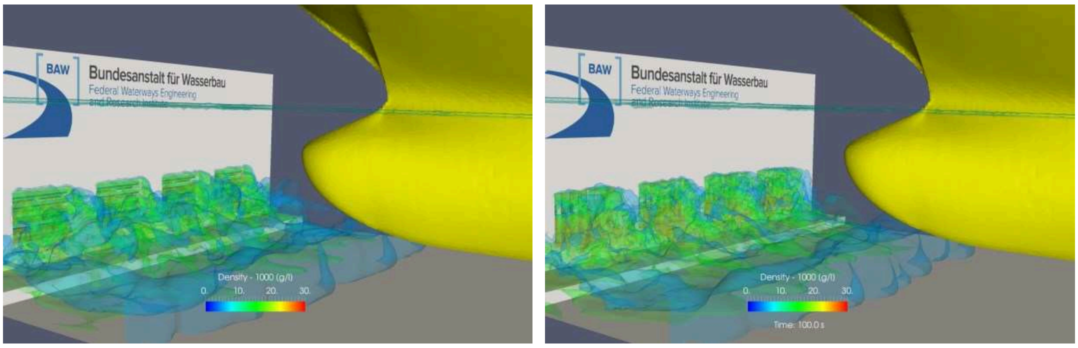


Figure 1: Salt water entering the model with horizontal breaker bars (left) and vertical breaker bars (right). The bow of the vessel is visible in yellow.

The solver "interDyMFoam" was enhanced for this task by a transport equation for salt with the diffusion coefficient coupled to the turbulence model. The density coupling was incorporated in the computation of the mixture density from the water content. Both a Boussinesq approximation and a full integration into the momentum equation were tested. The turbulence was computed on the basis of a Large Eddy Simulation (LES), using a k-equation subgrid-scale turbulence model. The large scale mixing of salt is thus directly modelled and the impact of stratification on the turbulence is directly taken into account. With this enhanced numerical model, the lock of Brunsbüttel was investigated. The model included moving valves and a moving vessel with six degrees of freedom, held in position by a set of springs which resembled the mooring line system. With this model, the impact of high salt contents and different geometries on the vessel was tested.

Literature

Thorenz, C.; Anke, J. (2013): Evaluation of ship forces for a through-the-gate filling system, Proceedings of the SMART RIVERS Conference 2013, Liege (BE), Maastricht (NL), 23-27 September 2013.

Examination of critical erosion velocities in spherical sediment with OpenFOAM

Authors: Arne Daniel, Leonie Walter, Jann Strybny University of Applied Sciences Emden/Leer

The beginning of sediment movement is of enormous importance for the durability of dredging operations. Highly empirical approaches are used for the description. The more exact consideration of the grain shape and the inclusion of the material composition (besides silicon dioxide also lime, wood and increasingly pollutants such as microplastics) would empirically overload the current approaches. Therefore, the group is working on strategies for grain-resolving modelling. The development of computing power suggests that in the next decade these approaches can be used to calculate hydraulically relevant areas of investigation. The first approach for a simulation to determine the beginning of particle motion is based on the hybrid model CFD-DEM (Computational Fluid Dynamics / Discrete Element Method). It consists in coupling open source algorithms of computational fluid dynamics (OpenFOAM®) with a simulation of the particle motion (LIGGGHTS®). Since physical as well as numerical prerequisites need to be met, the choice of the number of particles, geometric dimensions, initial and boundary conditions, turbulence and particle collision model, coupling properties and temporal and spatial resolution needs to be discussed carefully. Resulting values are compared with empirical findings from literature. The simulation was conducted with spherical particles of 2 mm diameter in a channel, initially resting layered in a hollow. The inlet velocity is increased from 0.1 to 0.6 m/s during the simulation to obtain a critical erosion velocity. Particle-particle collisions are modelled by a soft sphere model. Perfectly adapted experiments are to be conveyed in the future.

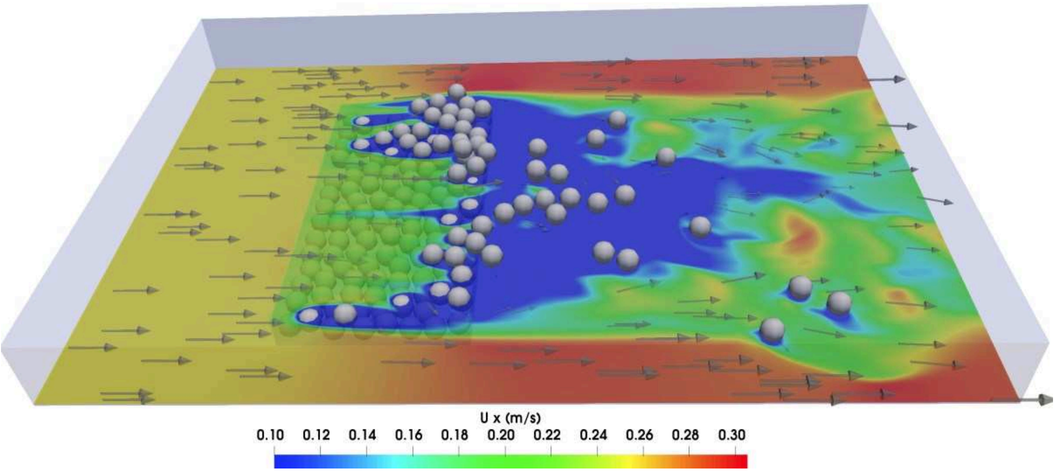


Figure 1: Simulation of spherical particles in a shear flow.

Literature

Kloss, C.; Goniva, C.; Hager, A.; Amberger, S.; Pirker, S. (2012): Models, algorithms and validation for opensource DEM and CFD-DEM. Progress in Computational Fluid Dynamics Vol. 12, pp 140-152.

Hager, A. (2014): CFD-DEM on Multiple Scales - An Extensive Investigation of Particle-Fluid Interactions. Ph.D. thesis. Johannes Kepler Universität Linz.

Federal Ministry
of Transport and
Digital Infrastructure

BAW
Federal Waterways Engineering
and Research Institute

Kussmaulstrasse 17 · 76187 Karlsruhe · Germany
Phone +49 (0) 721 9726-0 · Fax +49 (0) 721 9726-4540

Wedeler Landstrasse 157 · 22559 Hamburg · Germany
Phone +49 (0) 40 81908-0 · Fax +49 (0) 40 81908-373

www.baw.de