---

Conference Paper, Published Version

**Strybny, Jann; Thorenz, Carsten; Steinmill, Klaus; Schulze, Lydia; Hess, Max**

# On the numerical modeling of ship-waterway-interaction in canals

---

Vorgeschlagene Zitierweise/Suggested citation:
Strybny, Jann; Thorenz, Carsten; Steinmill, Klaus; Schulze, Lydia; Hess, Max (2012): On the numerical modeling of ship-waterway-interaction in canals. In: 10th International Conference on Hydroinformatics HIC 2012, July 14-18 2012, Hamburg, Germany. Hamburg: TuTech Verlag.

10th International Conference on Hydroinformatics
HIC 2012, Hamburg, GERMANY

# ON THE NUMERICAL MODELING OF SHIP-WATERWAY-INTERACTION IN CANALS

JANN STRYBNY (1), CARSTEN THORENZ (2), KLAUS STEINMILL (3),
LYDIA SCHULZE (2), MAX HESS (3)
*(1): Faculty of Maritime Sciences, University of Applied Sciences Emden/Leer, Bergmannstr. 36, 26789 Leer, Germany*
*(2): Federal Waterways Research Institute (BAW), Kußmaulstr. 17, 76187 Karlsruhe, Germany*
*(3): Hydraulic Engineering Laboratory, Georg-Simon-Ohm University of Applied Sciences, Keßlerplatz 12, 90461 Nuremberg, Germany*

Increasing sizes and velocities of vessels and limitations concerning the extensibility of inland navigation canals are leading to a substantial interest in modeling of ship-waterway-interactions. The following paper presents the investigation of the Open Source 3D Navier-Stokes-Solver “OpenFOAM” regarding its capability of modeling the ship-waterway-interaction. In hydraulic engineering practice, simplified and usable boundary conditions are necessary. The effort to set up a model must be reduced to a minimum. A set of boundary conditions which are relevant for hydraulic engineers is presented in the paper “On the numerical modeling of filling-emptying systems for locks” by *Thorenz et al.* at this conference. In this paper we present validation test cases for the methods shown in the aforementioned article.

## PREPROCESSING

### CAD applications

The modeling of real geometries of vessels, waterways and hydraulic structures can only be done efficiently, if external CAD-geometries are loaded into “OpenFOAM“. The complete chain of open source tools for modeling complex, realistic model geometries for ship-waterway-systems is presented (Figure 1.) In a first step, the modeling volume and a structured basic grid are defined in “OpenFOAM.” According to that, we recommend to continue preprocessing the external CAD-files with the GPL software called “Blender” (www.blender.org). Blender allows the merging of CAD-objects from different external sources and the definition of the correct location of a vessel geometry in a waterway geometry. In this way for example the adjustment of different drift angles can easily be made. Common Linux distributions include a tool called “Admesh.” “Admesh” exports an ASCII-stl-file witch can be interpretated by “OpenFOAM.” 3D objects constructed in CAD Software and exported for fluid dynamic purposes are often not exactly closed solids from a mathematical point of view. Solids saved in “Admesh” as a stl-file are “repaired” and fulfill the conditions required for mathematical modeling. Finally, the detailed disrcetization is done with the tool snappyHexMesh supplied by “OpenFOAM.”

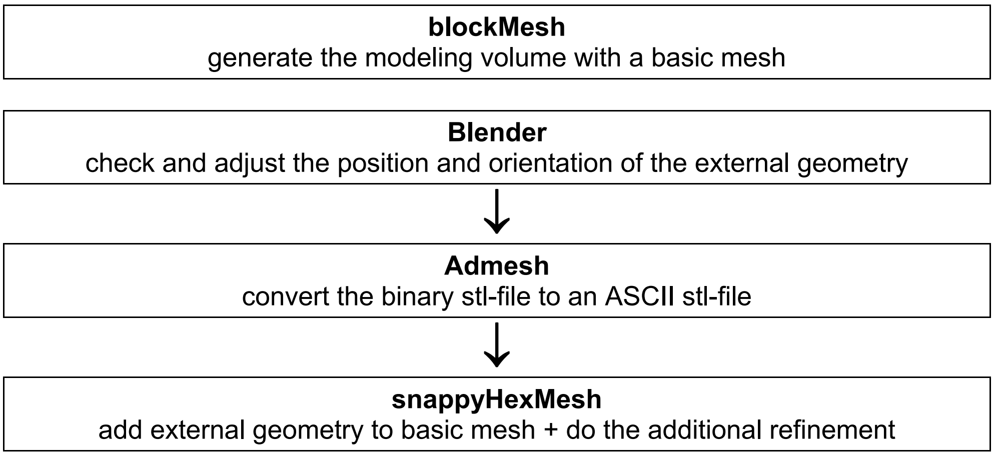

Figure 1. Chain of open source tools for doing the preprocessing

**Grid design**

The effect of VOF-strategies with an insufficient grid resolution on the accuracy of the determined surface evolution in the nearfield of hydraulic structures and vessels was investigated. It is generally accepted that VOF models need a high resolution and that this resolution has to be increased significantly in regions with high gradients such as free surfaces, close to the hull of vessels or hydraulic structures. Unfortunately, CFD practice shows either "extreme" resolution is not extreme enough or the increasing resolution is used in such a spacious way, that the number of elements is rising in the tens of millions range. The simulations become inaccurate or inefficient. In both cases, VOF methods are doomed to failure. For the further investigations we choose three different test cases: a vessel in a trapezoidal canal, an experimental lock and a weir with a flap on top.

Table 1. Resolution of three typical test cases discussed in this paper

| Type | Modeling volume l x w x h | max (Δx,Δy,Δz) | min (Δx,Δy,Δz) | $\min_{\text{Layer}}$ (Δx,Δy,Δz) | number of cells |
|---|---|---|---|---|---|
| lock | 12.5 m x 6 m x 13 m | 0.176 m | 0.022 m | - | 1 252 866 |
| weir | 29 m x 0.5 m x 7 m | 0.250 m | 0.0078125 m | 0.0003125 m | 406 589 |
| vessel | 320 m x 55 m x 8 m | 1.000 m | 0.250 m | 0.025 m | 1 513 409 |

Depending on the type of hydraulic structure, we have used a basic resolution between 0.176 m and 1.000 m. This basic grid complies with the requirements in greater distance to the object of investigation and of modeling the air space above the water surface. Based on several test cases, we came to the result that regions with high gradients have to be resolved in at most four levels of refinement starting from the chosen basic grid. We advise to refine the grid only in a grid design process with simultaneous consideration of fluid dynamic processes approximately expected in the different regions of the modeling volume.

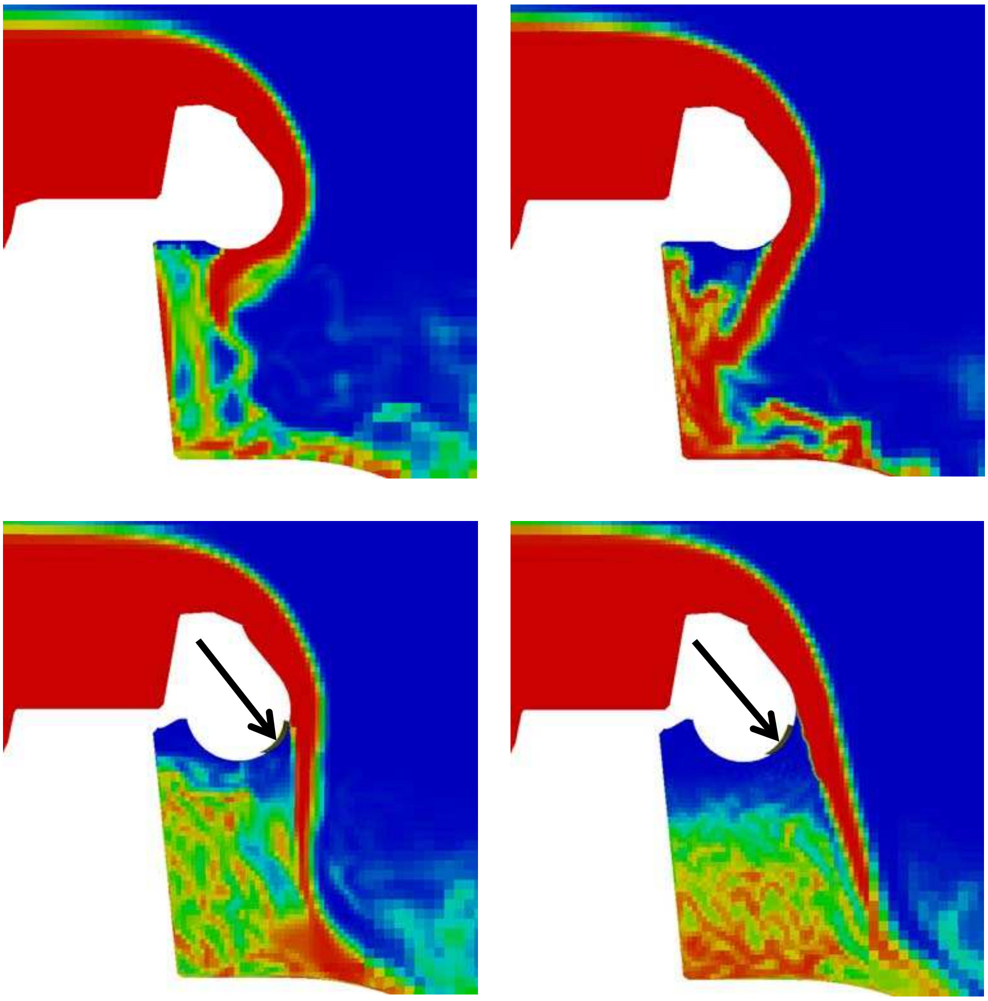

Figure 2. Separation of a jet influenced by the grid resolution (red: α=1. blue: α=0.)

This procedure proved to be particularly difficult while modeling a weir with a flap on top. This case is discussed in detail in Figure 2. Close to the water surface and the bottom, a grid refinement of one level was done (0.125 m.) In the region of the hull of the weir and the following stilling basin, a refinement of two levels (0.0625 m) and close to the separation point, a refinement of three levels (0.03125 m) was chosen (Figure 3, left image.) The result of a calculation in Figure 2 top left shows a distinct suction process making an extreme refinement of the wall boundary layer obligatory. “OpenFOAM” offers a function for the subsequent insertion of highly refined wall boundary layers. Generating a refined boundary layer and defining aeration zones left and right to the suction zone only has a low influence (Figure 2 top right.) The location of an aeration boundary directly on the hull at the lee side of the weir (Figure 3 arrow) causes a separation of the jet that exactly reaches the upper edge of the aeration vent (Figure 2 down left.) Consequently, the separation of the jet is not a result of the natural hydrodynamics of the weir, but it would be influenced by the modeler via positioning the aeration vent along the hull of the weir.

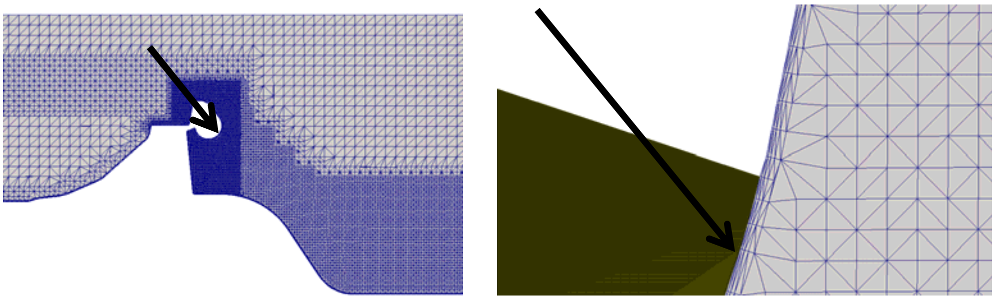

Figure 3. Vertical slice of the discretization of a weir

In order to reproduce a physically correct separation of the jet, air has to get in a small tapered slot and the flow field obligatory for this effect has to be resolved with an adequate number of elements. A plausible separation resulted after the wall boundary layer in the region of the separation slot contained elements with a minimized grid space (Figure 3 right image.) Only the exact reproduction of the real aeration system would help to avoid the extreme refinement. This example shows that a VOF simulation of such processes is possible if detailed knowledge of the physics of the flow field and the estimated scales influences the grid design from the beginning. The calculation time will be prolongated by the tiny grid sizes, but a number of cells that got out of control could be avoided by a well-directed refinement.

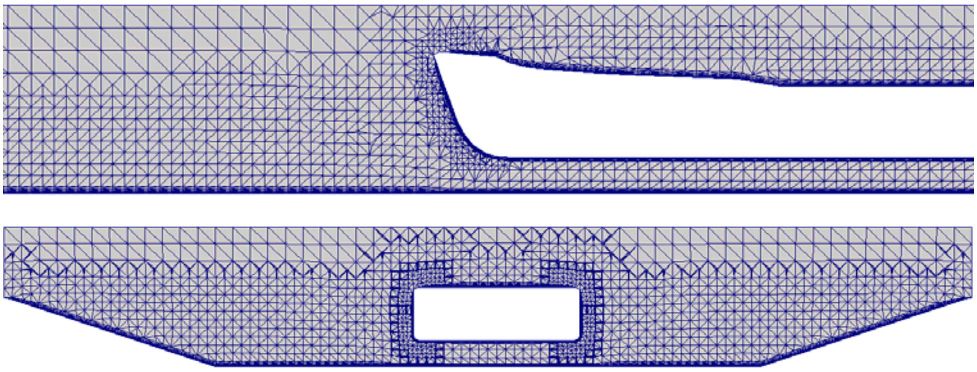

Figure 4. Vertical slices of the discretization close to the vessel in a trapezoidal canal

Regarding the system of a vessel in a canal, the grid design was easy to realize on the basis of the previous experiments. The basic grid in the canal and the air space was resolved with 1.000 m. The water volume in the investigation zone was resolved with a grid space of

0.250 m. The bottom of the canal and the wall boundary layer of the vessel contain elements down to 0.025 m. The test case was resolved with a moderate grid that leads to acceptable, not optimal results. Only in this way can the simulations be reproduced by users (students) with affordable calculation resources.

## TURBULENCE MODELING

Turbulence modeling is a crucial point to succeed in 3D modeling of vessels, waterways and hydraulic structures. Regarding turbulence, our focus is on Large-Eddy type models only. In scales less than the grid size, the principle of the different LES modifications is defined by the subgrid model. Besides the simple Smagorinsky-approach and a number of modified approaches more or less close to the ideas of Smagorinsky, subgrid models of higher value are including a transport equation. “OpenFOAM” provides one equation subgrid models on the basis of a k-model (“OpenFOAM” key word: oneEqEddy) and one equation subgrid models with a transport equation for the eddy viscosity as recommended by Spalart and Allamaras.

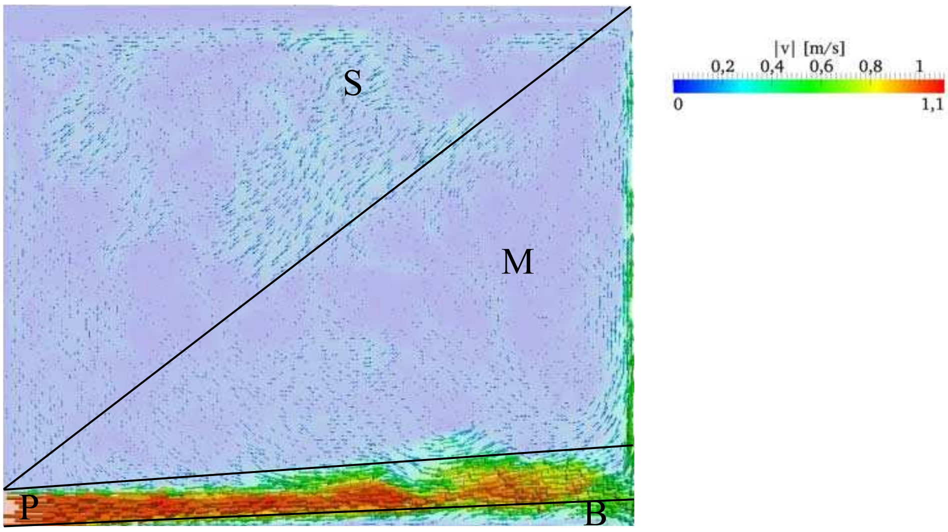

Figure 5. Experimental box - mixing processes of a nozzle jet modeled with LES + One Equation subgrid model

Using the example of a simplified box and an experimental lock, the different LES concepts are verified against empirical formulas (Bollrich [1]) regarding the enlargement of nozzle jets. The investigation leads to the result that LES concepts with one equation subgrid models outperform the Smagorinsky subgrid models. The different zones of a nozzle jet (primary zone (P), bottom boundary zone (B), mixing zone (M), secondary zone (S)) can be reproduced with a qualitatively and quantitatively convincing result. Regarding the reliability of any other “OpenFOAM” application, LES with a one equation subgrid model according to Spalart Allamaras (“OpenFOAM” key word: SpalartAllamarasDDES) become widely accepted in our simulations.

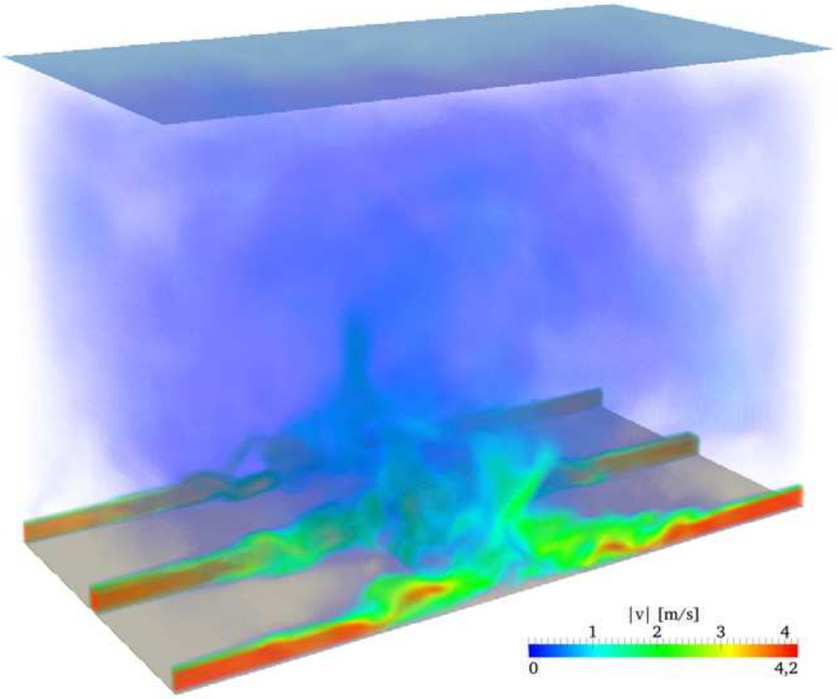

Figure 6. Experimental lock: mixing processes of nozzle jets modeled with LES + One Equation subgrid model, visualized with “Paraview”

## SIMULATION OF THE PRIMARY WAVE

### Pressure field

For the determination of the maximal sunk of the primary wave besides the vessel, it is more efficient to reconstruct the surface elevation on the basis of the pressure field. Because of modeling a quasi steady state, the flow field is free of local acceleration. The vessel is located in a prismatic canal. Besides the highly shaped bow and stern areas, the flow field is nearly free of advective acceleration. For this reason, it is permissible to gather the surface elevation directly from the pressure. The accuracy of the VOF-concept does not lead to the calculation of an acceptable primary wave on the basis of a coarse grid with a minimum grid space of 0.250 m. Secondary waves are not visible on the surface due to the fact that the wave length is too short in comparison to the chosen grid space. A wave length has to be resolved with a minimum of 10 elements. Further tests have shown that the resolution in space has to be heightened by a factor of 4 to determine the sunk with the VOF concentrations as well.

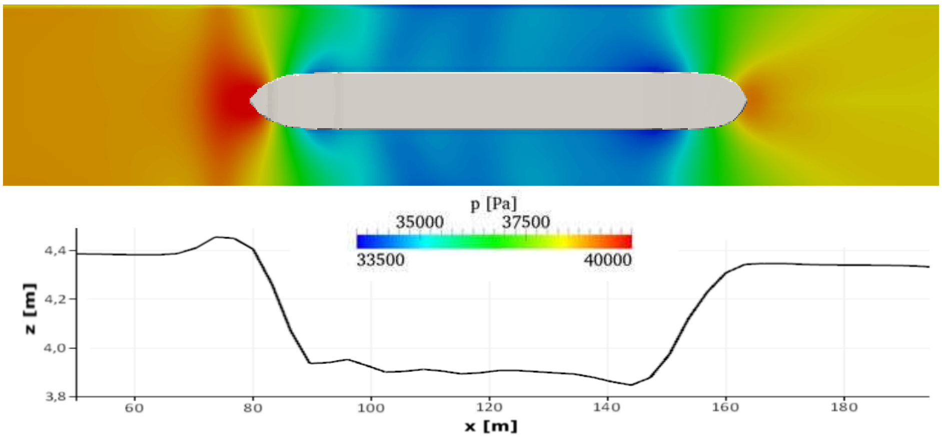

Figure 7. Pressure field and reconstructed surface elevation besides the vessel

**Verification**

To evaluate the advantages and disadvantages of the numerical model and a corresponding laboratory model and for optimizing the validation process itself, both types of models were built by the same team. For validation of the numerical simulation, the data from the laboratory model is more precise compared to data from field testing because inaccuracy in the model can be determined and measuring can be reproduced. In this specific case, the advantage of the laboratory model is the ability to avoid all environmental influences like wind or uneven ground, which would occur in the field.

Figure 8. Geometry of the numerical model and the laboratory model

The available laboratory flume was suitable for the construction of a 1:25 scaled standard trapezoidal canal (real width of 55 m), which is the most common type for navigable inland canals in Europe. The model was equipped with a 1:25 model of a Johann-Welker type vessel and systems for contactless measurement of surface elevation and the velocity of the vessel. The dependency between velocity and squat is supposed to provide reference values for the later validation of the "OpenFOAM" calculations. The verification provided convincing results. With a view to an economic processing, the minimum grid space could be limited to 0.250 m (with the exception of the boundary layer). An increasing refinement would lead to a further decreasing variance. Furthermore, the deviation between the numerical results and the laboratory results of about 10 percent has to be evaluated in view of the accuracy of available measurement techniques. The measurement of surface elevations within the range of 1 mm is accepted as standard in hydraulic engineering laboratories. However, the limitations of measured electronic signals must be taken into consideration. It is assumed that the laboratory measurement and the numerical model exceed the field measurement in providing a trapezoidal canal and a centered navigation of the vessel from a mathematical point of view. On the other hand, the increased sunk measured in nature is a result of the propulsion unit of the vessel inducing a momentum on the hull. Neither in the numerical model nor in the laboratory model are these effects considered.

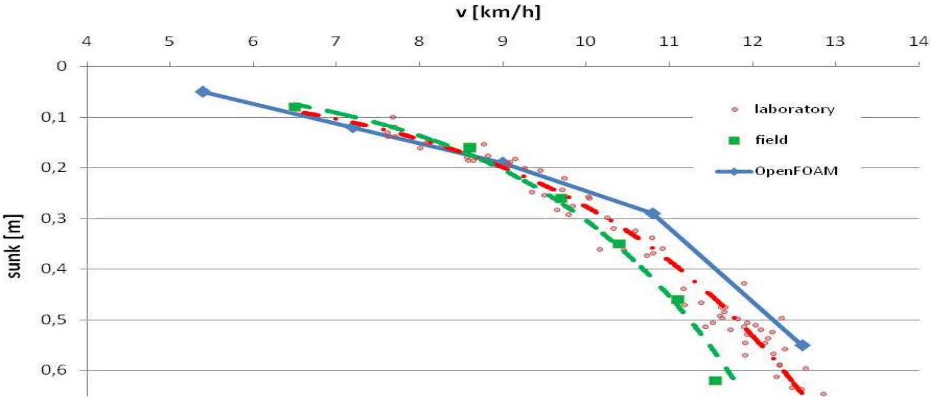

Figure 9. Comparison of the maximum sunk  (centered vessel in a trapezoidal canal )

The typical examples to become acquainted with “OpenFOAM” are called “Breaking of a Dam,“ “obstacle” or “tank” with highly simplified geometries and a highly reduced number of involved physical processes. The enormous diversity of implemented processes, boundary conditions etc. associated with a limited documentation make it a time-consuming procedure for the user, adapting “OpenFOAM” accurately to realistic hydraulic engineering structures like locks, weirs, vessels or offshore-structures. This was the motivation to document “OpenFOAM” regarding the gamut of hydraulic engineering applications. A user- and benchmark manual including these detailed and reproducible hydraulic engineering case studies will be presented. On a new web-domain www.hydroscience.de, the manuals and “OpenFOAM” input files will be made publicly available after the conference.

## REFERENCES

[1] Bollrich, J. et al.: „*Technical Fluid Dynamics Volume 2 – Spezial Problems*“. pp 237 – 323, in German, Verlag für Bauwesen, Berlin, 1989.

[2] Strybny, J., Thorenz, C., Croce, R., and Engel, M., “A parallel 3d free surface navier-stokes solver for high performance computing at the german waterways administration”. *Proc. 7th Int. Conf. on Hydroscience and Engineering*, Philadelphia, USA, 2006.

[3] Wendling, I.:”*Dynamic Large-Eddy-Simulation of turbulent flow in the vicinity of complex geometries*”. in German, Fluid Dynamics Section, TU Darmstadt, 2007.