11th International Conference on Hydroscience & Engineering
"Hydro-Engineering for Environmental Challenges"
September 28th - October 2nd 2014
Hamburg, Germany

Mini-Symposium
CFD in the Nearfield of Structures

In the proximity of hydraulic structures and vessels, there is a highly three-dimensional interaction of the fluids water and air. Buildings are over and under flown and vessels are immersed into the fluid. Hydraulic jumps, water jets, breaking waves and free flying water are leading to situations with multiple free surfaces along a perpendicular axis. Standard 3D models for river hydraulics are not necessarily in the position to describe these phenomena. For the approximation, a two-phase Navier-Stokes solver is indispensable.

A large number of commercial software for multi-phase flow is available. This software is optimized for the aircraft and automobile industry. The interests of clients coming from the domain of hydraulic engineering are often not adequately represented. Commercial software being closed source makes own developments and extensions to the software difficult or impossible. Any extension must be commissioned at high cost.

The open source CFD code "OpenFOAM®" attracts worldwide attention in a broad range of engineering applications. But 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 of the mini-symposium to discuss "OpenFOAM®" regarding the gamut of hydraulic engineering applications. Topics like

- fluid dynamics close to weirs or hydro-electric power plants
- filling-emptying systems of locks
- ship-waterway-interaction
- flow forces
- sediment transport in the vicinity of hydraulic structures
- coastal engineering topics like wave generation, wave breaking and wave runup
- design of offshore-structures
- greenshipping

are leading to a substantial interest in modeling on the basis of OpenFOAM®. We are deeply interested in outstanding abstracts on this field.

Contact:
Dr.-Ing. Carsten Thorenz, Federal Waterways Engineering and Research Institute, Karlsruhe,
carsten.thorenz@baw.de

Prof. Dr.-Ing. Jann Strybny, Faculty of Maritime Sciences, University of Applied Sciences Emden/Leer,
jann.strybny@hs-emden-leer.de