12th International Conference on Hydroscience & Engineering
Hydro-Science & Engineering for Environmental Resilience
November 6-10, 2016, Tainan, Taiwan.

High-Resolution Numerical Analysis of Flow Over a Ground Sill Using OpenFOAM

Katharina Teuber[1], Tabea Broecker[1], Matthias Barjenbruch[2], Reinhard Hinkelmann[1]
1. Chair of Water Resources Management and Modeling of Hydrosystems, TU Berlin
2. Chair of Urban Water Management, TU Berlin
Berlin, Germany

ABSTRACT

In the field of hydraulic engineering, attention towards Computational Fluid Dynamics (CFD) has increased within the last years. In this study, flow over a two-dimensional ground sill is simulated and analyzed using the open source model OpenFOAM. Single-phase flow simulations are compared to experimental results obtained by Almeida et al. (1993) and two-phase flow simulations are compared to analytical solutions by using Bernoulli's and continuity equation. The results show that the model is capable of simulating such hydraulic testcases. Different RANS and LES simulations were found to reproduce the analyzed flow behavior well.

KEY WORDS: OpenFOAM, CFD, interFoam, two-phase flow, fluid mechanics

INTRODUCTION

Within the last years, Computational Fluid Dynamics (CFD) has gained importance in the field of hydraulic engineering. Several publications, such as Bayón-Barrachina et al. (2015 a), Bayón-Barrachina & López-Jiménez (2015 b), Schulze & Thorenz (2014) and Thorenz & Strybny (2012) have investigated complex hydraulic testcases such as hydraulic jumps, and filling and emptying of locks using the open source model OpenFOAM. In these publications the Volume of Fluid (VoF) approach for two-phase flows has been used in order to describe free surface flows. But the model also offers the possibility to simulate cases where two phases are of importance. One application is the simulation of in-sewer processes (Edwini-Bonsu & Steffler (2004), Gessner et al. (2014), Hvitved-Jacobsen et al. (2013)). In this work, a first step of the validation process regarding two-phase flows in closed ducts such as sewer pipes is made. The model is used to simulate flow over a two-dimensional ground sill. Validation is first performed by comparing the results of single-phase flow with measurements by Almeida et al. (1993). Two-phase flow simulations are performed for different two- and three-dimensional model setups, variations are made concerning the structure of the sill, discharge, water level and the flow regime.

COMPUTATIONAL FRAMEWORK

Numerical model

Surface water flow is calculated by using the two-phase flow solver interFoam based on a volume of fluid (VoF) approach for one- and two-phase flows. Both phases are considered as one fluid with rapidly changing fluid properties, therefore one set of Navier-Stokes-equations is solved. The phases are distinguished by an additional transport equation for the volume fraction which is used as a marker to describe the distribution of the phases throughout the domain. The equations can be formulated as follows (Rusche, 2002):
Mass conservation equation:

$$\nabla \cdot \vec{U} = 0 \tag{1}$$

Momentum conservation equation:

$$\frac{\partial \rho \vec{U}}{\partial t} + \nabla \cdot (\rho \vec{U}\vec{U}) = -\nabla prgh + \nabla \cdot (\mu \nabla \vec{U}) + (\nabla \vec{U}) \cdot \nabla \mu - \vec{g} \cdot \vec{x} \nabla \rho \tag{2}$$

where prgh is the static pressure minus hydrostatic pressure:

$$prgh = p - \rho g h \tag{3}$$

Volume of Fluid equation:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \vec{U}) + \nabla \cdot ((1-\alpha) U_r \alpha) = 0 \tag{4}$$

with the following parameters:

$$\rho = \alpha \rho_w + \rho_a (1-\alpha) \tag{5}$$

$$\mu = \alpha \mu_w + \mu_a (1-\alpha) \tag{6}$$

where $\vec{U}$ is the velocity field [m/s]; ρ is the density [m³/s]; t is time [s];

p is pressure [Pa]; μ is dynamic viscosity [Ns/m²]; $\vec{g}$ is acceleration vector due to gravity [m/s²]; $\vec{x}$ is a spatial position vector [m]; α is volume fraction or indicator function [-]; U_r is the relative velocity between the phases [m/s]; the subscripts a and w denote different fluids air and water.
The indicator function α is defined as:

$$\alpha = \begin{cases} 1 & \text{fluid w} \\ 0 < \alpha < 1 & \text{transitional region} \\ 0 & \text{fluid a} \end{cases} \qquad (7)$$

For single-phase simulations the volume fraction α is 1 and constant over the whole domain and during the simulation time.

Turbulence modelling

Turbulence effects and their impact on the flow have been simulated using different Reynolds averaged (RANS) turbulence models and Large Eddy Simulations (LES). From the wide range of RANS models the Standard k-ε (Launder & Sharma, 1974), k-ω (Wilcox, 1988) and k-ω Shear Stress Transport (SST) model (Menter, 1993, 1994) were used. As subgrid scale model for the LES simulations the Smagorinsky model (Smagorinsky, 1963) was chosen.

Boundary conditions

For all cases presented in this paper, a similar set of boundary conditions has been used. The inlet of the domain has been subdivided in two components, an inlet for the air phase and an inlet for the water phase. The height of the water inlet depends on the desired water level and the flow is prescribed by using a fixed flow velocity or a discharge. The air inlet is specified by using a total pressure boundary condition. The upper and lower walls of the domain are defined using a no-slip condition. The outlet of the domain is specified by using a pressure boundary condition. The water level in the domain is fixed by defining a weir in close proximity to the outlet as outlined in Bayón-Barrachina et al. (2015 a). For three-dimensional testcases, the sidewalls are determined using a no-slip condition. Two-dimensional testcases are simulated with so-called empty boundary conditions which are implemented in OpenFOAM to describe sidewalls of a two-dimensional geometry.

Geometry and mesh

Unstructured meshes with local refinements at the walls were set up using the open source mesh generation tool gmsh. The single-phase flow cases (Figure 1) consist of 12,106 cells for the RANS turbulence models and 89,284 cells for the LES simulations, leading to a cell length between 0.0042 m and 0.056 m for the RANS simulations and between 0.00035 m and 0.041 m for the LES simulations.

The two-dimensional model domains of cases 1 and 2 consist of 68,542 cells with a minimum cell length of 0.0024 m and a maximum cell length of 0.15 m. The three-dimensional setup is based on the two-dimensional domain but extended in z-direction by ten layers. The model consequently consists of 685,420 cells. The domain of case 3 consists of 175,762 cells. The cell length ranges between 0.0035 m and 0.144 m.

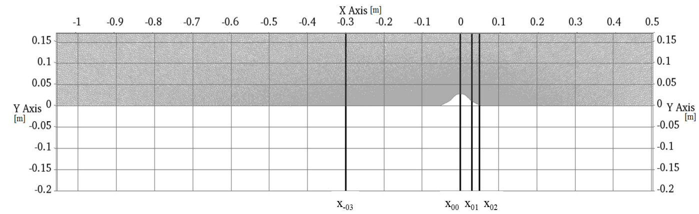

Figure 1. Model setup and measurement locations of single-phase case

SINGLE-PHASE FLOW

In order to analyze the accuracy of the interFoam solver regarding flow behavior behind a two-dimensional ground sill, a single-phase testcase has been implemented using different turbulence models. Experimental data for this case has been obtained by Almeida et al. (1993) and is available in the database of the European Research Community on Flow, Turbulence and Combustion (ERCOFTAC) (Davroux et al., 1995). The domain consists of a two-dimensional duct bounded by an upper and lower wall with a polynomial shaped obstacle on its bottom. The mean centerline velocity at the inlet amounts to 2.147 m/s. At four, respectively two different locations the velocities in x- and y-direction were compared to the experimental results: x_{-03} = -0.30 m (in front of the sill, inlet profile), x_{00} = 0.00 m (top of the sill), x_{01}= 0.03 m (end of the sill), x_{02}=0.05 m (recirculation zone). Turbulence models used were the RANS and LES models previously outlined.

The results show that the chosen RANS models as well as the LES simulations lead to a good approximation of the experimental results (Figure 2 and Figure 3), however, the LES simulations are able to capture fluctuations as well which can be of interest when analyzing eddy structures behind ground sills. One disadvantage of LES is the higher resolution of the mesh that is needed in order to display large scale eddies which lead to much higher computation times.

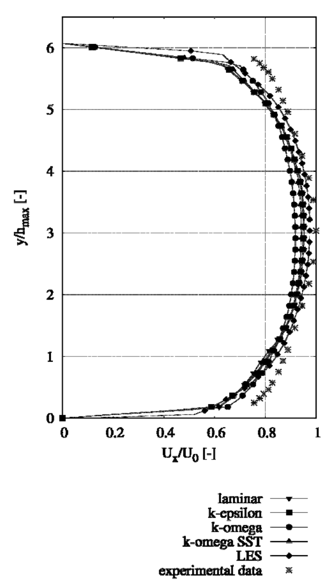

Figure 2. Velocity profiles at x_{-03} = -0.30 m (inlet profile)

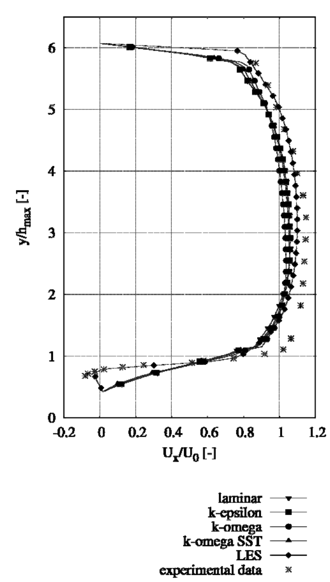

Figure 3. Velocity profiles at x_{01} = 0.03 m (end of sill)

TWO-PHASE FLOW

When analyzing two-phase flow over a two-dimensional ground sill, two main aspects are of interest: eddy structures behind the sill and the water level drawdown. Since the accuracy concerning eddy structures has already been analyzed in the previous case, the focus of the two-phase flow cases will lie on the water level drawdown. Since the water level drawdown is constant as soon as a quasi steady-state is reached, LES simulations are not necessary. Therefore, the Standard k-ε-turbulence model has been chosen in this part in order to save computation time (approximately 1 h instead of 4 h for parallel computation on 16 processors).

As a first step, the effect of the sill structure on the water level drawdown has been analyzed. A two-dimensional model setup similar to case 1 as listed in Table 2 has been used. Angular shaped structures with three different angles for the ground sill as well as a round structure have been investigated (see Table 1). The maximum height of the sill is Δz = 0.20 m and similar for all cases.

Table 1. Overview over different sill structures

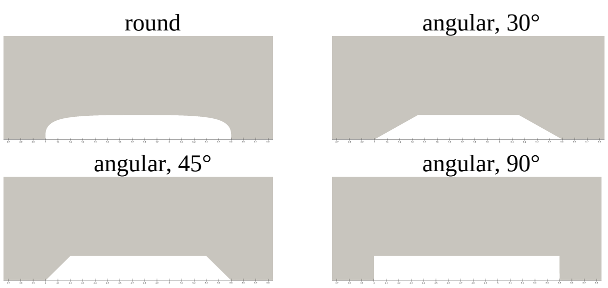

Figure 5 shows the resulting water level drawdowns for the different structures. The maximum water level drawdown is the smallest (Δh = 0.03 m) for the round structure. The drawdown for the 30° and 45° angular structure are in a comparable order of magnitude (Δh = 0.04 m). A significantly higher drawdown is caused by the ground sill with an angle of 90° (Δh = 0.09 m). With an analytically calculated drawdown of Δh = 0.036 m (Table 2), the sill structures with small angles or round structures show the highest accuracy. The effect of the ground sill is higher for steeper hill structures which can be explained by higher single losses for steeper structures.

Table 2. Subcritical two-phase flow: Properties of different testcases

	Case 1	Case 2	Case 3	Case 4
Length of domain	25 m	25 m	35 m	25 m
Height of domain	2 m	2 m	6 m	2 m
h_1	1.0 m	1.0 m	3.0 m	0.3 m
v_1	1.00 m/s	1.25 m/s	3.00 m/s	3.00 m/s
Δz	0.2 m	0.2 m	0.2 m	0.1 m
Δh, analytical	0.036 m	0.070 m	0.110 m	0.100 m
Δh, numerical	0.042 m	0.090 m	0.140 m	0.055 m

In a next step, changes have been made concerning the water depth and the flow velocity using the 30° angular structure of the ground sill. All cases have subcritical flow conditions and do not show a flow transition over the ground sill.

The setups including the analytically calculated water level drawdown and the simulated water level drawdown are listed in Table 2 (case 1 to 3), where v_1 is the approaching velocity and h_1 the water depth in front of the sill. The results show a good agreement of the simulated results with the analytical solution obtained by using Bernoulli's and continuity equation. A reason for the slightly higher drawdown obtained by the numerical solution are additional energy losses which have already been shown when analyzing the influence of the structure due to the structure of the sill.

Case 1 has also been extended to a three-dimensional geometry with a width of 1 m and sidewalls with no-slip condition.

The resulting water level drawdown is compared to the drawdown resulting from the two-dimensional simulation in Figure 6. The figure shows that the three-dimensionality of the testcase causes a smaller drawdown and a shorter length of the water level drawdown. The reason for this is the influence of the sidewalls. The no-slip condition at the sidewalls causes additional continuous energy losses that were neglected in the two-dimensional geometry.

As a next step, a strictly supercritical setup (case 4) has been analyzed. The supercritical flow case has been simulated with a height of the ground sill of Δz= 0.10 m and an approaching flow velocity v_1 = 3 m/s (see Table 2). Compared to the analytical rise of the water level (Δh = 0.10 m), the maximum increase achieved in the simulations (Δh = 0.055 m) is considerably smaller. The high difference between analytical and numerical solution can be considered reasonable since the high flow velocity of this testcase leads to higher single losses at the sill structure and therefore a higher deviation between analytical and numerical solution.

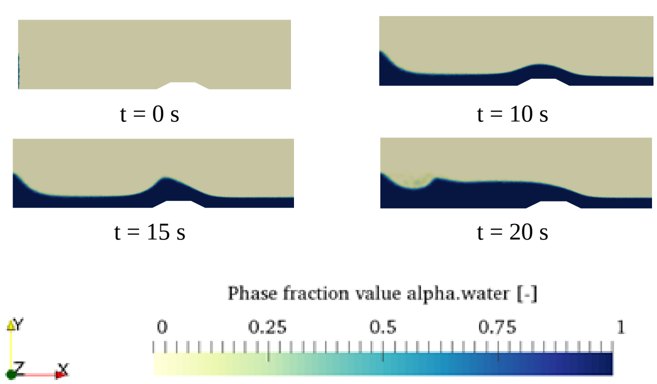

Figure 4. Filling of the domain, segment of the computational domain for different time steps

In the last case, the stability of the simulations has been analyzed under initially dry conditions. A water level of h_1 = 1 m and a flow velocity of v_1 = 1m/s has been chosen at the inlet (similar to case 1) and at the outlet a free outflow without weir. Figure 4 shows the behavior of the two phases in the domain at different time steps. After a simulation time of 20 seconds a quasi-steady state is reached and subcritical conditions in the upstream part of the domain and supercritical conditions in the downstream part of the domain are reached. A flow transition occurs over the ground sill. Close to the inlet, a disturbance of the water surface can be found. This disturbance develops due to an eddy in the air phase that is caused by the increase of the water level from the ground sill upstream to the inlet. Due to the inlet boundary patch the eddy is trapped in this point. This effect could be moved further upstream by choosing an inlet in a higher distance to the ground sill.

CONCLUSIONS

In this study, flow over a two-dimensional ground sill has been analyzed using OpenFOAM.

First, a single-phase flow has been simulated and different turbulence models have been validated concerning their accuracy in describing the eddy structure behind the sill. All models analyzed led to a good accuracy, however, the LES turbulence model was capable to account for fluctuations as well. One disadvantage of the model is the smaller necessary grid size which leads to much higher computation times.

In a second step, two-phase flow has been simulated using the k-ε turbulence model and different parameters such as the structure of the ground sill, discharge, water level and flow regime have been evaluated concerning their influence on the water level drawdown.

Due to additional energy losses caused by the angular structure of the sill as well as losses due to sidewalls when a three-dimensional model is chosen, the analytically calculated drawdown is smaller than the two-dimensional simulations but reasonably coincides for different simulations. Changes of the water level drawdown due to the structure of the ground sill and sidewalls when three-dimensional testcases are computed are plausible. The simulations are also stable for a filling case with initially dry conditions.

Summing up, the VoF approach implemented in OpenFOAM is capable of describing flow over a ground sill and similar hydraulic cases. Future research aims to closer look at the behavior of the air phase.

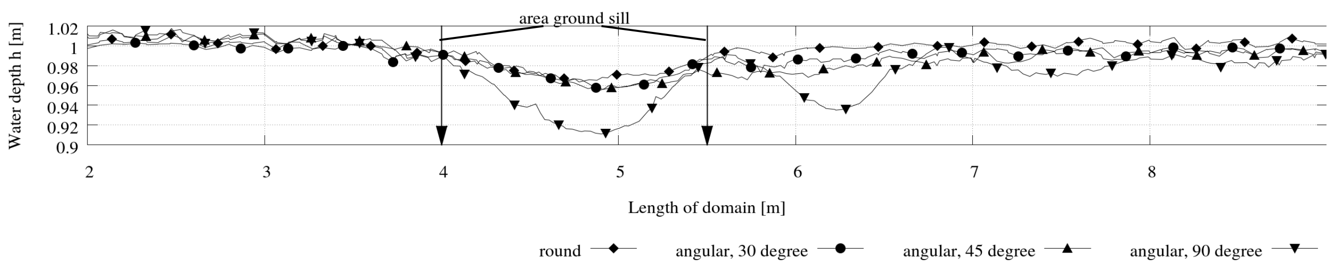

Figure 5. Comparison of water level drawdown for different sill structures

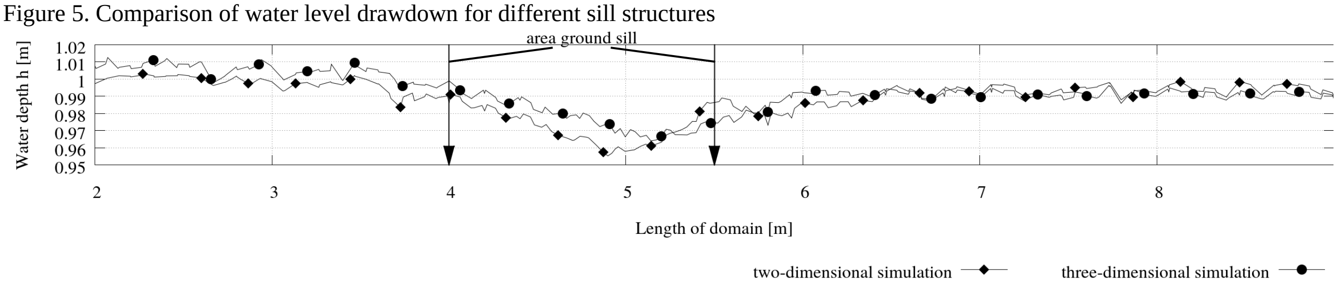

Figure 6. Comparison of water level drawdown for two- and three-dimensional model setup

ACKNOWLEDGEMENTS

This work is carried out within the DFG Research Training Group Urban Water Interfaces which is a joint initiative of TU Berlin and Leibniz-Institute of Freshwater Ecology and Inland Fisheries (IGB) Berlin. The authors acknowledge the funding.
Parts of the simulations were computed on the supercomputers of Norddeutscher Verbund für Hoch- und Höchstleistungsrechnen in Berlin.

REFERENCES

Almeida, G., Durao, D. and Heitor, M. (1993). Wake flows behind two-dimensional model hill. Experimental Thermal and Fluid Science, 7(1): 87-101.

Bayón-Barrachina, A., Vallés-Morán, F. J. & López-Jiménez, P. A. (2015 a). Numerical analysis and validation of South Valencia sewage collection system diversion. 36th IAHR World Congress, The Hague, The Netherlands.

Bayón-Barrachina, A. & López-Jiménez, P. A. (2015 b). Numerical analysis of hydraulic jumps using OpenFOAM. Journal of Hydroinformatics, 4 (17), 662-677.

Davroux, A., Hoa, C. & Laurence, D. (2014). Flow Over a 2D Hill- Reference Solutions for k-ε and Second Moment Closure Turbulence Models. 4th ERCOFTAC Workshop on Turbulence Modelling, Karlsruhe, Germany.

Edwini-Bonsu, S. & Steffer, P. M. (2004). Airflow in sanitary conduits due to wastewater drag: a computational fluid dynamics approach. Journal of Environmental Engineering Science, 3 (5), 331-342.

Gessner, M. O., Hinkelmann, R., Nützmann, G., Jekel, M., Singer, G., Lewandowski, J., Nehls, T. & Barjenbruch, M. (2014). Urban water interfaces. Journal of Hydrology, 514, 226-232.

Hvitved-Jacobsen, T., Vollertsen, J. & Nielsen, A. H. (2013). Sewer processes: microbial and chemical process engineering of sewer networks (Vol. 2), CRC press.

Launder, B.E. & Sharma, B.I. (1974). Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc. Letters in Heat and Mass Transfer, 1 (2), 131{137.

Menter, F.R. (1993). Zonal two-equation k-ω turbulence models for aerodynamic flows. AIAA Journal, p.2906.

Menter, F.R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32 (8), 1598-1605.

Rusche, H. (2002) Computational fluid dynamics of dispersed two-phase flows at high phase fractions, PhD thesis, Imperial College London (University of London), United Kingdom.

Schulze, L. & Thorenz, C. (2014). The Multiphase Capabilities of the CFD Toolbox OpenFOAM for Hydraulic Engineering Applications. Proceedings of the 11th International Conference on Hydroscience and Engineering (ICHE), Hamburg, Germany.

Smagorinsky, J. (1963). General circulation experiments with the primitive equations: I. the basic experiment*. Monthly weather review, 91 (3), 99-164.

Thorenz, C. & Strybny, J. (2012). On the numerical modelling of filling-emptying systems for locks. Proceedings of the 10th International Conference on Hydroinformatics (HIC), Hamburg, Germany.

Teuber, K., Broecker, T., Elsesser, W., Agaoglu, B., Hinkelmann, R. (2016). Investigation of flow over a ground sill using OpenFOAM. CMWR 2016, Toronto, Canada.

Wilcox, D. C. (1988). Re-assessment of the scale-determining equation for advanced turbulence models. AIAA Journal, 26 (11), 1299-1310