6th IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, September 9-11, 2015, Ljubljana, Slovenia

INVESTIGATION OF DIFFERENT METHODS FOR MODELLING INTERACTION BETWEEN HYDROKINETIC RIVER TURBINES AND WATER SURFACE

Christa STADLER*

University of Stuttgart, Institute of Fluid Dynamics and Hydraulic Machinery, Stuttgart, Germany

Stefan RIEDELBAUCH

University of Stuttgart, Institute of Fluid Dynamics and Hydraulic Machinery, Stuttgart, Germany

ABSTRACT

For numerically modelling the interaction between hydrokinetic river turbines and the adjacent water surface of the river flow a preliminary study on the treatment of open channel flow is meaningful. Special requirements on meshing and defining boundary and initial conditions for this specific task are described. Appropriate methods and solvers in OpenFOAM® and Ansys CFX are tested and evaluated through comparison with different experimental results from literature. An in-house solver based on the shallow water equations is also able to solve complex open channel problems with some limitations. Deviations from experimental results are observed for flows with high gradients in water surface and hence a high component of vertical flow velocity, which is by definition neglected within this approach. Comparing the simulations using two-phase models with experimental results some deviations occur for decelerating flow and in regions close to walls. All in all good solutions are archived for all 2-phase solvers which therefore may be used for modelling of kinetic turbines close to the water surface.

KEYWORDS

Hydrokinetic River Turbine, Open Channel Hydraulics, Two-Phase Flow Models

1. INTRODUCTION

Providing 16% of the global electric energy supply hydropower holds a substantial contribution to the renewable energy sources. But, only about 35% of the estimated worldwide economic potential on hydropower is currently used for power production [1] – most of it by conventional damming methods. But there is also a worldwide high potential for the use of alternative facilities extracting energy from watercourses, e.g. axial hydrokinetic river turbines.

During the last years several of these turbine types were developed at the Institute of Fluid Mechanics and Hydraulic Machinery of Stuttgart University. In order to protect rotor and generator but also to increase the power output the developed turbines types are shrouded and feature a diffuser downstream of the runner. Due to the cubic relation between power output and axial velocity installation sites providing high free flow velocities are required. Another limiting factor regarding the site is the minimum flow depth. This value is presently

* *Corresponding author:* Institute of Fluid Dynamics and Hydraulic Machinery, University of Stuttgart, Stuttgart, Germany, phone: +49 71168560235, email: christa.stadler@ihs.uni-stuttgart.de

determined by the influence of the bottom boundary layer in river flows but also by a buffer zone to the water surface in case of low water discharge or surface waves. Up to now complex open channel phenomena like stationary waves are not taken into account concerning those considerations. Fig.1 shows the results for a first test simulation approaching the problem of interaction between water surface and kinetic turbines. Downstream of the turbine v-shaped surface waves develop and the upstream water level is piled up by the resistance of the turbine in the flow.

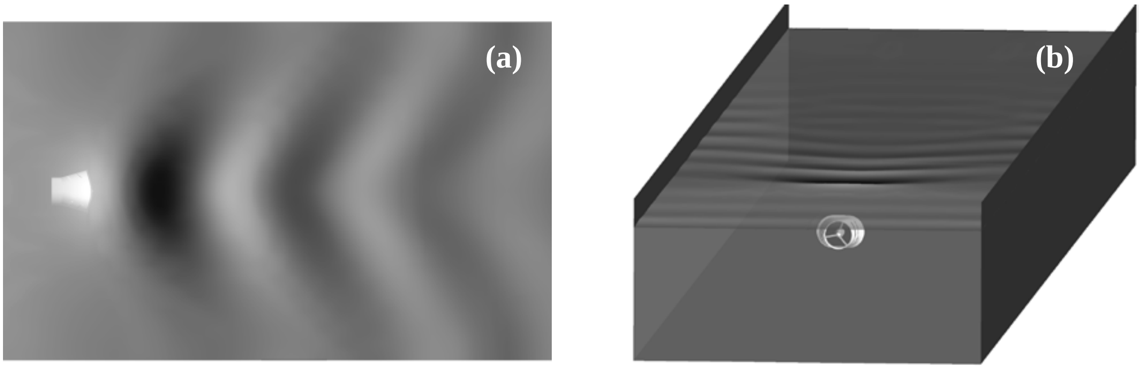

Fig.1Interaction between river turbine and water surface in a channel: top view (a) and 3D view (b)

Similar behaviour was observed by [2] for an experimental setup investigating a 1:30 model of an unshrouded tidal turbine in a test channel with free surface flow. For a detailed numerical investigation of this phenomenon preliminary studies on appropriate modelling procedures and methods are necessary being the objective of the current work.

2. FUNDAMENTAL THEORY

For an expedient case setup in numerical modelling of open channel flow the knowledge on relevant hydraulic theory as well as on the attributes of the different available solvers is important, as boundary and initial conditions, but also physical and mathematical models have to fit to the problem.

Theory on open channel hydraulics

The relative energy head in a channel is – according to Bernoulli's theorem – a function of flow depth h, discharge Q and channel width b and may be calculated with:

$$H = h + \frac{Q^2}{2gh^2b^2} \qquad (1)$$

This formula is a first order approximation and only holds for the assumption of slightly curved stream lines und water surface as discussed in detail in [3]. Rearranging Eq. (1) for h three solutions are mathematically possible: one of them is physical invalid (e.g. negative), the other two solutions are physically correct which is pointing out to two possible flow regimes for one energy head in open channel flows. This may be described by taking a look at the dimensionless Froude number, which is the most significant characteristic number for free surface flows. It is defined by the ratio of speed of wave propagation and the flow velocity and is calculated as:

$$Fr = \frac{v}{\sqrt{gh}} \qquad (2)$$

with v being the mean velocity and h being the characteristic average flow depth in corresponding cross sectional area. By means of the Froude number open channel flows may be classified in 2 regimes: subcritical and supercritical flow.

For $Fr < 1$ the observed flow is in subcritical (streaming) regime, where gravitational effects are dominating the flow characteristics. In this regime the information on relevant flow variables is transported both in and against the flow direction. When flow is dominated by the flow velocity, conditions for a supercritical (shooting) regime with $Fr > 1$ are fulfilled [4]. Due to the relatively high flow velocity upstream flow variables are not influenced by the downstream flow.

Within a transitional flow critical flow conditions are reached where $Fr = 1$ and the flow depth is explicit. The change in flow regime may be triggered by a change in the channel cross section (e.g. blockage by a kinetic turbine), bed slope or bed roughness. A rapid transition from sub- to supercritical flow is called 'hydraulic drop' and the reverse case is called 'hydraulic jump' and occurs for an instant change from super- to subcritical flow which means that the water depth before the jump is always lower than behind [4]. Depending on the upstream Froude number and the ratio of up- and downstream flow depth up to 85% of the hydraulic energy may be dissipated in hydraulic jumps. Therefore it is important to exclude the possibility of developments of hydraulic jumps caused by a hydrokinetic turbine in its direct vicinity.

Theory on numerical treatment of open channel flow

<u>The 2D depth averaged solver tidalFoam</u>: The in-house solver tidalFoam developed by [5] is an extension of shallowWaterFoam which is a basic solver within the open source code OpenFOAM®. The solver is based on the shallow water equations where the flow velocity is depth averaged and its vertical component is neglected. Unlike the original solver variables for the channel bathymetry and Manning roughness are implemented in tidalFoam. Furthermore two turbulence models (Smagorinsky- and k-ε-model), a model with a source term for kinetic turbines and a wetting/drying-algorithm are available.

<u>The Volume of Fluid (VoF) method</u>: the VoF-method is a common approach for modelling multiphase flow. Both the liquid fluid in the channel (water) and the ambient gaseous fluid (air) are taken into account. The flow fields are shared by the two fluids. Due to the shared velocity field no slip between the water and the air phases occurs. In order to determine the region of the phase interface an additional flow field α is introduced. This variable indicates the volume fraction of the phase "water" in a cell using a step function which then is one for a cell filled with water and zero for cells filled with air. For post processing the water surface is commonly defined at a volume fraction $\alpha = 0.5$. The motion of the volume fraction is determined by solving an additional transport equation. Ansys CFX uses an interface capturing approach with a compressive discretization scheme in time and space to reduce smearing at the phase interface [7]. InterFoam uses a similar interface compression algorithm for sharpening the phase interface.

<u>The inhomogeneous multiphase Model in CFX</u>: In contrast the VoF-method only the pressure field is shared by air and water for the inhomogeneous multiphase model. For the other flow variables individual transport equations for each phase have to be solved which significantly increases the computational time. At the interface the flow fields of the different fluids are connected with interphase transfer terms which directly depend on the surface area of the interface. Within this work the free surface model provided by CFX is used for the interphase treatment [6]. Furthermore within the inhomogeneous model a homogenous turbulence field is applied.

3. REMARKS ON MESHING, BOUNDARY AND INITIAL CONDITIONS

Caused by the implicit character of flow depth for a given energy head in open channels (see chapter 2) but also by the complex interface between water and air numerical models describing open channel flows have to be carefully defined. Regarding the mesh for two-phase model setups the following problem occurs: as shown in Fig. 2 (a) using a nearly orthogonal mesh may cause small artificial surface waves at the water-air-interface, whereas for a mesh with the same number of cells but a refinement normal (see Fig. 2 (b)) to the water surface those waves do not occur. This also increases the accuracy as, according to [7], the smearing of the interface reaches over two to three cells in Ansys CFX. As it is using a similar interface capturing approach this also applies to interFoam.

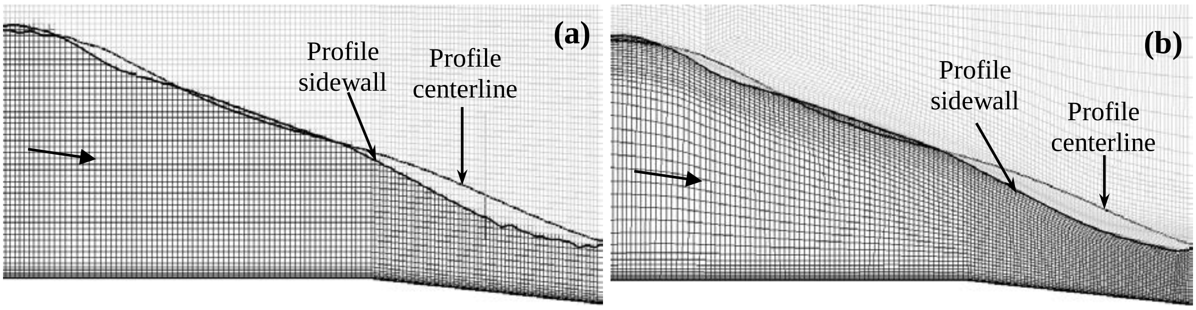

Fig.2 Water surface profiles with mainly orthogonal (a) and manually adapted (b) mesh

A recommended procedure used for the realized simulations is to run a first simulation using a mesh which is not specially structured and in a second step adapting and refining the mesh normal to the water surface (cell height: 2mm; growth ratio: 1.1) gained from the result of the initial simulation except in regions where unsteady free surface movement is present. For the following validation cases the water surface is expected to be steady state. Hence, a manual adaption provides a higher mesh quality and a shorter runtime.

BC	two-phase models			tidalFoam	
	variable	subcritical	supercritical	subcritical	supercritical
Inlet	Velocity **U**	$Q /(h_{us}(t_{i-1})\, b)\, \alpha$	$U\, \alpha$	**hU**	**hU & h**
Outlet	Pressure **p** or **p_rgh**	$\rho\, g\, (h_{ds} - y)\, \alpha$ $\rho\, g\, h_{ds}\, \alpha$	all zero gradient	**h**	all zero gradient

Tab.1 Variables for different flow regimes at in- and outlet boundaries

Special requirements are set on the boundary and initial conditions for open channel flow, mainly at the in- and outlet. The boundary conditions have to reproduce the physical flow conditions as listed in Tab.1. For supercritical flow the upstream water level has to be fixed whereas for subcritical flow it is part of the solution. Therefore, the inlet velocity has to be recalculated every time step according to the continuity equation to ensure a constant discharge Q. This approach is based on the 2D wave absorption described in [8]. Up to now this is only implemented for outer iteration loops of both solvers. The disadvantage of this method is that steps in the sequence of $Q(t)$ and hence in the upstream water level $h_{us}(t)$ may occur due to insufficient mesh resolution in the interphase area.

Regarding the downstream water level h_{ds} a hydrostatic pressure distribution must be defined when subcritical regime is expected. For supercritical regime the pressure at the outlet is set to zero gradient in streamwise direction. For all boundaries the volume fraction α is set to zero gradient. Defining the boundary and initial conditions different from the intended regime may lead to a numerically converging but incorrect solution as exemplarily described in the following: setting a fixed water level or hydrostatic pressure at the downstream boundary for an intended supercritical flow a hydraulic jump will develop right before the outlet. This is a numerically valid solution but does not reflect reality. For all two-phase solvers the SST-k-ω turbulence model is used, whereas for simulations in tidalFoam turbulence is modelled with the k-ε-model. The basic versions of the refined meshes provide a dimensionless wall distance of $y^+ < 120$, which according to [9] is sufficient for the chosen turbulence model. For the transient simulation an adaptive time step allowing a maximum Courant number of 0.2 is used.

4. TEST CASES

In order to evaluate different solvers and models being potentially capable of solving open channel flow problems, different experimental setups from literature are used as validation cases. Suitable setups are quasi steady open channel flow cases were both flow regimes occur and a change in flow depth is forced by a change in cross section, as it is the case for a hydrokinetic turbine in a channel. Appropriate circumstances may be found e.g. in Venturi channel flows usually used as discharge measurement devices or weir overflows. The main criterion is the comparison of water surface profiles, but also velocity profiles are compared, if available. The following test cases are simulated for validation:

Transitional flow through a modified Venturi channel (Case I)

The first test case is a flow through a modified Venturi channel experimentally and analytically investigated by H. W. Hager in 1985 [3]. As illustrated in Figure 3 a vertical cylinder is positioned in the centre of a test rig to locally reduce the cross section. A discharge of $Q = 0.02$ m³/s leads to subcritical flow in the upstream region. Downstream of the obstacle the flow is not piled-up and hence supercritical flow is observed. In the area of the critical cross section (minimum channel width) a hydraulic drop with highly curved streamlines and large gradients in surface slope occurs due to the instant reduction of channel width.

Fig.3 Schematic illustration of the Setup for Case I with post processing cross sections (white)

Simulations are set up according to chapter 3. Resulting surface profiles for time averaged solutions using different solvers are presented in Fig. 4 (a). The solution for the flow in the region from inlet to critical cross section is steady over time. The upstream flow depth and the water level in the critical cross section are nearly identical for all two-phase solvers, which are collectively underestimating the experimental results by about 2.3%. Using a mesh with $y^+ < 5$ for the simulations the upstream water level increases by about 0.6%.

Due to effects correlating to a von Kármán vortex street the wake flow of the cylinder shows highly transient behaviour. Here the different methods and models are significantly influencing the results. Fig. 4 (b) shows the surface profile in a downstream cross section for different time steps using the interFoam solver. The water surface is greatly varying over time, with a main maximum oscillating left and right and up and down in the cylinder wake. The time averaged flow depth shows a good compliance with the experimental results which are measured via point gauge and are therefore not high precise in this transient region. Using the interFoam solver also simulations with the k-ε turbulence model are performed but no major differences in flow are detected compared to the SST-k-ω-model. Evaluating the downstream water surface profile for the VoF-method and the inhomogeneous model in

Ansys CFX it is noteworthy that both approaches overestimate the maximum height of the wake flow.

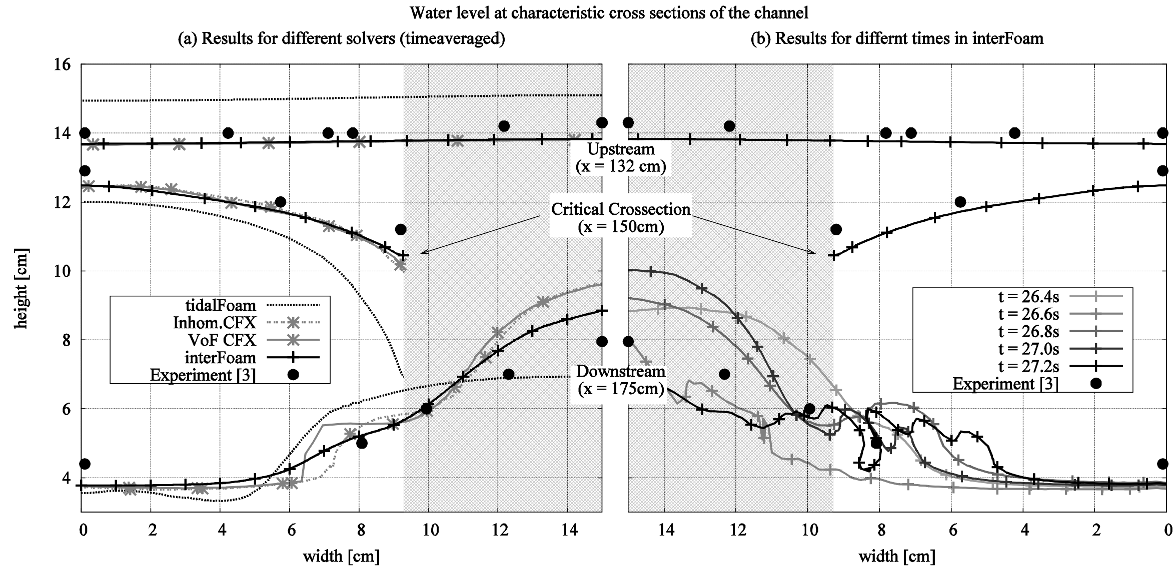

Fig.4 Water surface profile at characteristic cross sections for Case I: time averaged results for different solvers (a) and different times in interFoam (b)

The deviations of the tidalFoam solver in the critical cross section result from high gradients in longitudinal flow velocity over flow depth which are observed for the two-phase simulations and for the experiment and cannot be described by the depth averaged shallow water equations. Especially in the direct vicinity of the cylindrical obstacle flow velocity is overestimated by tidalFoam which triggers a major underestimation of flow depth there. This error also propagates upstream and results in an increased head water level.

Transitional flow through a classical Venturi channel (Case II)

For this test case a classical Venturi channel with a mutual narrowing along the side walls and a comparatively long section of constant constricted width is investigated. The side wall contour is schematically shown in Fig 5 (b). In the expansion area at the rear end of the Venturi channel a downward ramp is installed changing the level of bottom geometry. For a discharge of $Q = 0.1164$ m³/s a subcritical flow develops upstream of the constriction, piled-up by the reduction of cross section.

The flow regimes in and behind the Venturi channel are similar to *Case I*, but the streamlines and water surface are only slightly curved due to the moderate channel contour. During the experiment Engelund and Munch [10] observed a stationary surface cross wave pattern within this area which is directly depending on the Froude number of the flow.

In Fig. 5 results are presented by surface profiles in different cross sections (a) and in a longitudinal cut along the centre of the channel and along the channel side wall (b). Comparing the VoF solvers slight deviations are visible. It is observed that the intersection points between surface profile at the centreline and sidewalls gained from the experiment and from the two-phase solvers are identical. Consequently, this approach is capable of modelling the correct wave length and hence imitates the experimentally observed pattern of surface waves in the channel. Deviations between the two-phase simulations and the experiments are observed mainly at the channel side wall where the position of the water surface is underestimated by the numerical model. The solvers also show inaccuracy for modelling the decelerated phase of flow as may be seen in the results for cross section $x = 1.2$ m in Fig 6 (a).

A comparison of the homogenous (VoF) and the inhomogeneous approach in Ansys CFX provides the exact same results in water surface position for both methods.

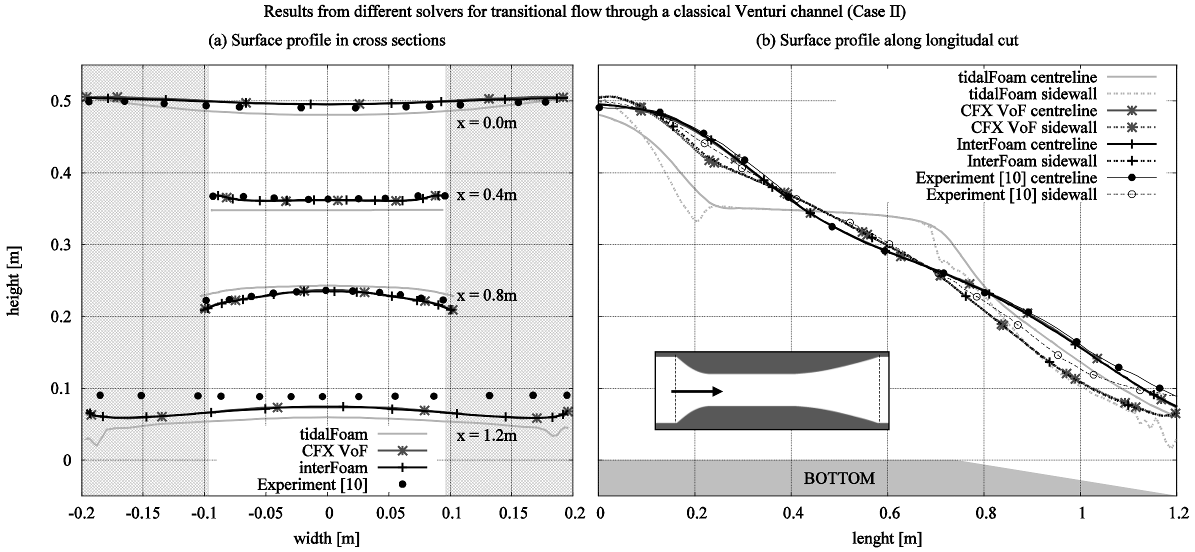

Fig.5 Water surface profiles results for different for classical Venturi channel: solvers at different cross sections (a) and longitudinal surface profiles (b)

Regarding the solutions gained from the tidalFoam simulations in Fig. 6 (b) the water level along the longitudinal cut is nearly constant where the channel width is constant. This test case shows that for flow with high gradients in surface profile, the depth averaging of the flow velocity is inappropriate as the flow is dominated by vertical velocity. This component is missing by definition in the 2D depth averaged shallow water equation.

Supercritical flow downstream of a classical Venturi channel (Case III)

For the third test case a Venturi channel very similar to *Case II* is investigated, however the flume is in supercritical regime all the way from inlet to outlet. Due to the expansion of channel width at the rear end of the Venturi channel cross waves develop as qualitative pictures from experimental and numerical test rig illustrate in Fig 6 (a) and (b). These surface waves reside in a completely steady state.

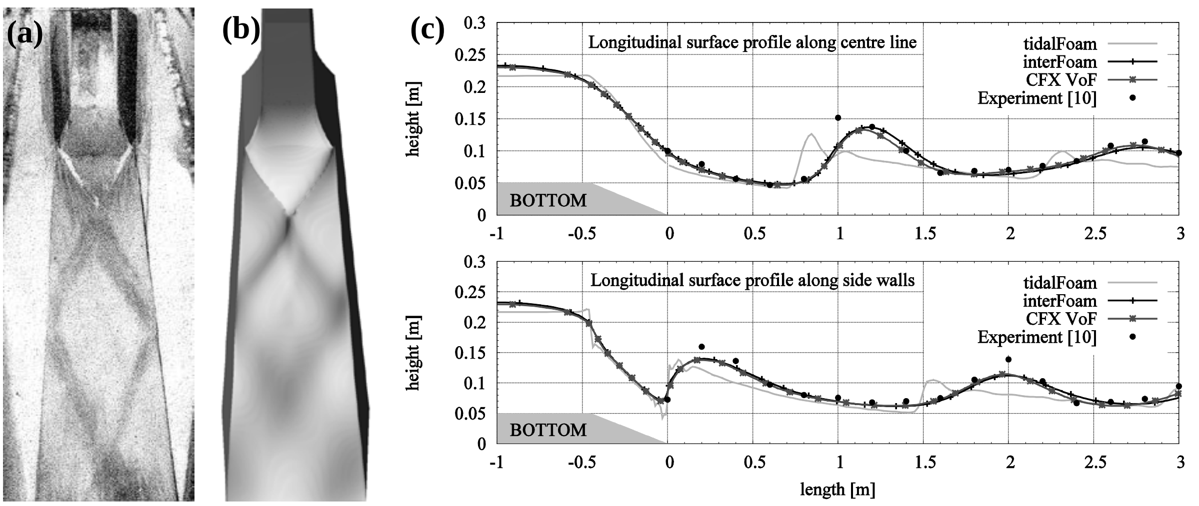

Fig.6 Photography of water surface during experiment [10] (a), Isosurface for a corresponding VoF-simulation (b) detailed surface profiles for different solvers (c)

Comparing the centre line of the water surface profile along flow direction the two-phase simulations show a good agreement with the experimental results. The only exception is the first interference in the centre of the channel where the experimental water level is remarkably higher. For the water surface profile at the channel side wall higher deviations are observed mainly for the peaks of surface waves which are determined too low by the two-phase solvers. As already observed for *Case II* the homogenous and the inhomogeneous model provide similar results also for the present *Case III.*

For this case tidalFoam provides fairly reasonable results considering the fact that flow fields are depth averaged. The deviations to the two-phase solvers and the experimental results are mainly observed within the shorter wavelength of the downstream surface waves. These result from the negligence of the vertical velocity component as already observed in *Case I* and *II*: the slope of the water surface is calculated too steep by the solver. The wavelength of the downstream wave pattern is determined too short due to an indirect proportionality between Froude number and wave length described in detail by [10].

5. CONCLUSION

Different solvers and methods are compared with regard to their capability of simulating open channel flow phenomena. The special requirements on numerical modelling of open channel flow problems regarding physically correct boundary and initial conditions are shown.

Within the validation of different methods good results are achieved for all two-phase solvers. Deviations between the VoF-method and the inhomogeneous approach (requiring an extremely large computational effort) are only observed for flow with breaking water surface. Some deviations occur for decelerated flows and for the determination of water levels at wall boundaries. The 2D shallow water solver tidalFoam is sufficient for a first approach to the flow condition if gradients in flow velocity and water surface are moderate. For complex flow transitioning between the two flow regimes inaccuracies may occur. Also for a detailed determination of the flow field around kinetic turbines a 3D method is required. Several further investigations are planned to improve the accuracy of these models.

6. REFERENCES

[1] Horlacher, H.-B.: Globale Potenziale der Wasserkraft. *Expertise for the WBGU Report 'World in Transition: Towards Sustainable Energy Systems'. Springer-Verlag*. 2003

[2] Myers, L.; Bahaj, A. S.: Wake studies of a 1/30th scale horizontal axis marine current turbine. *Ocean Engineering*. Vol 34. No. 5. 2007. pp. 758-762.

[3] Hager, W. H.: Modified Venturi channel. *Journal of irrigation and drainage engineering*. Vol. 111. No. 1. 1985. pp. 19-35.

[4] te Chow, V.: Open channel hydraulics. *McGraw-Hill Book Company, Inc*., New York. 1959

[5] Ruopp, A., Daus, P., Ruprecht A., Riedelbauch S.: A Two-Dimensional Finite Volume Shallow Water Model for Tidal Current Simulations Using OpenFOAM® - Numerical Validation and High-Resolution Ocean Modelling Case. *European Wave and Tidal Energy Conference (EWTEC),* Aalbourg, Denmark, 2013

[6] ANSYS Inc.,: ANSYS CFX Solver Theory Guide, Release 15, 2013

[7] Svihla, C. K.; Xu, H.: Simulation of free surface flows with surface tension with ANSYS CFX. *2006 International ANSYS Conference*. Pittsburgh. 2006.

[8] Higuera, P.; Lara, J. L.; Losada, Inigo J.: Realistic wave generation and active wave absorption for Navier–Stokes models: Application to OpenFOAM®. *Coastal Engineering*. Vol 71. 2013. pp. 102-118

[9] Menter, F. R.; Kuntz, M.; Langtry, R. Ten years of industrial experience with the SST turbulence model. *Turbulence, heat and mass transfer*. Vol. 4. No.1. 2003

[10] Engelund, F.; Munch-Petersen, J.: Steady flow in contracted and expanded rectangular channels. *La Houille Blanche*. Vol. 4. 1953. pp. 464-481.