River Flow 2010 - Dittrich, Koll, Aberle & Geisenhainer (eds) - © 2010 Bundesanstalt für Wasserbau ISBN 978-3-939230-00-7

Numerical investigations of free surface flow in a channel with a Long contraction

Hans Bihs
Department of Hydraulic and Environmental Engineering, NTNU Trondheim, Norway

ABSTRACT: In the present study a three dimensional numerical model is used to predict the flow in a channel with a long contraction and a ne gravel bed. The numerical results are compared with experimental data. In the beginning of the contraction the water accelerates. A distinct drop in the water level can be observed. In the expansion area a recirculation zone forms. Here the water elevation increases again, but to a lower level than before the contraction. For the cases with a higher discharge, a local hydraulic jump occurs in the beginning of the contraction. The numerical results show good agreement with the experimental data. The numerical model uses the level set method in addition with Lagrangian particle correction for the calculation of the free water level. With this front-capturing method the free surface is modeled as the zero level set of a scalar signed distance function. In order to maintain this property and to ensure mass conservation, the level set function is reinitialized after each time step. Surface tension is taken into account with the continuum surface force method. The convective terms including the level set function and the equations of the k-ω turbulence model are discretized with the fifth-order finite difference WENO scheme. It ensures a smooth and oscillation free solution for large gradients and even shocks while maintaining a high order discretization at the same time. The pressure is discretized with the projection method. The Poisson equation for the pressure is solved with the preconditioned BiCGStab algorithm. The staggered grid configuration leads to a tight velocity-pressure coupling. For time discretization a second-order Adams-Bashforth scheme is used. Parallelization of the numerical scheme is achieved by using the domain decomposition framework together with the MPI library. Since the numerical model employs a Cartesian grid, the present study requires special attention to the geometry of the contraction and the expansion. Here an immersed boundary method based on ghost cell extrapolation is used.

Keywords: Free surface flow, Particle level set method, Immersed boundary

1 INTRODUCTION

In many engineering disciplines fluid flows with more than one phase occur. The correct prediction of the phase interface is important for many applications, such as solidification, melt dynamics, reacting flows, breaking surface waves and air-water dynamics. Several interface capturing approaches exist. One of the first was the Marker-and-Cell (MAC) scheme [5]. Here massless marker particles are used to represent the phases. The distribution of the particles determines the location of the interface, which must be reconstructed explicitly. The computing effort is rather large because the grid needs to be refined along interface in order to avoid smeared solutions. In the Volume of Fluid method (VOF) [7] the marker particles are replaced by a scalar field, which describes the volume fraction of one fluid for each discretization cell. The main difficulty with this method is the accurate reconstruction of the interface. Also an effect known as foaming, the smearing of the interface can be a problem. In the current paper the level set method is used [10]. The main idea behind this method is that the location of interface is represented implicitly by the zero level set of the smooth signed distance function. In contrast to the VOF method the level set function varies continuously across the interface. The location of the interface is readily available and does not require any reconstruction procedure. Since mass conservation is not enforced directly by the level set method, it is supplemented with a particle correction algorithm.

2 NUMERICAL MODEL

2.1 *Equations of Motion*

For the investigations in the present paper a three-dimensional numerical model is used. The governing equations for the mass and momentum conservation are the continuity and the incompressible Reynolds-averaged Navier-Stokes (RANS) equations:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \overline{u_i u_j}\right] + g_i \tag{2}$$

U is the velocity averaged over the time t, x is the spatial geometrical scale, ρ is the water density, ν is the kinematic viscosity, P is the pressure, g is the gravity, u is the velocity fluctuation over time with $\overline{u_i u_j}$ representing the Reynolds stresses. At solid boundaries the surface roughness is accounted for by using Schlichting's rough wall law [12].

2.2 *Turbulence Model*

Turbulence is modeled with the k-ω model by Wilcox[18]. The Reynolds stress term in the the RANS equations is then replaced with the Boussinesq-approximation:

$$-\overline{u_i u_j} = \nu_t\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) - \frac{2}{3}k\delta_{ij} \tag{3}$$

with

$$\nu_t = c_\mu \frac{k}{\omega} \tag{4}$$

The turbulent kinetic energy k and the specific turbulent dissipation ω which are needed to determine the eddy viscosity ν_t are obtained by solving the following transport equations:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + 2\nu_t |S|^2 - k\omega \tag{5}$$

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + 2c_\mu c_{\omega 1}|S|^2 - c_{\omega 2}\omega^2 \tag{6}$$

A slight deviation between the above presented equations for the k-ω model and Wilox's model is that here ω is c_μ times Wilcox's. That way the eddy viscosity representation resembles that of the standard k-ε model. The model coefficients are then c_μ = 0:09, $c_{\omega 1}$ = 5=9, $c_{\omega 2}$ = 5=6 and $\sigma_\omega = \sigma_k$ = 2. The term $|S|^2$ is constituted of the mean rate of strain tensor

$$S_{ij} = \frac{1}{2}\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right) \tag{7}$$

Wall functions according to [18] are used to model the surface roughness of solid boundaries.

2.3 *Level Set Method*

The level set method was first presented by Osher and Sethian [10] in 1988. It was devised for computing and analyzing the motion of an interface Γ between two phases in two or three dimensions. The location of interface is represented implicitly by the zero level set of the smooth signed distance function $\phi(\vec{x},t)$. In every point of the modeling domain the level set function gives the closest distance to the interface and the phases are distinguished by the change of the sign. This results in the following properties:

$$\phi(\vec{x},t)\begin{cases} > 0 \; if \; \vec{x} \in phase\,1 \\ = 0 \; if \; \vec{x} \in \Gamma \\ < 0 \; if \; \vec{x} \in phase\,2 \end{cases} \tag{8}$$

Also the Eikonal equation $|\nabla\phi| = 1$ is valid. When the interface Γ is moved under an externally generated velocity fieeld $\vec{v}$, a convection equation for the level set function is obtained:

$$\frac{\partial \phi}{\partial t} + U_j \frac{\partial \phi}{\partial x_j} = 0 \tag{9}$$

When the interface evolves, the level set function looses its signed distance property. In order to maintain this property and to ensure mass conservation the level set function is initialized after each time step. In the present paper a PDE based reinitialization equation is solved [14]:

$$\frac{\partial \phi}{\partial t} + S(\phi)\left(\left|\frac{\partial \phi}{\partial x_j}\right| - 1\right) = 0 \tag{10}$$

S(ϕ) is the smoothed sign function by Peng et. [11].

$$S(\phi) = \frac{\phi}{\sqrt{\phi^2 + \left|\frac{\partial \phi^2}{\partial x_j}^2\right| (\Delta x)^2}} \tag{11}$$

With the level set function in place, the material properties of the two phases can be defined for the whole domain. Without special treatment there is a jump in the density ρ and the viscosity ν across the interface which can lead to substantial numerical stability problems. The solution is to define the interface with the constant thickness 2ε. In that region smoothing is carried out with a regularized Heavyside function H(ϕ). The thickness ε is proportional to the grid spacing, in the present paper it was chosen to be ε = 1:6Δx. The density and the viscosity can then be written as:

$$\begin{aligned} \rho(\phi) &= \rho_1 H(\phi) + \rho_2 (1 - H(\phi)) \\ \nu(\phi) &= \nu_1 H(\phi) + \nu_2 (1 - H(\phi)) \end{aligned} \tag{12}$$

and

$$H(\phi) = \begin{cases} 0 & if\ \phi < -\varepsilon \\ \frac{1}{2}\left(1 + \frac{\phi}{\varepsilon} + \frac{1}{\Pi}\sin\right) & if\ |\phi| < \varepsilon \\ 1 & if\ \phi > \varepsilon \end{cases} \tag{13}$$

2.4 *Lagrangian Particle Correction*

In order to improve the mass conservation properties of the original level set method, massless particles are used to correct the level set function in under-resolved regions. This approach was first presented in [4]. In the present paper an improved version of this algorithm is used [17]. Negative particles are seeded on the negative side of the level set function in a narrow band of 1:6Δx near the interface, the positive particles are respectively placed on the positive side of the interface. Each cell in the narrow band contains 64 particles. The particles are advected with the third-order accurate TVD Runge-Kutta scheme [13]. In contrast to the original method, the narrow band in which the particles are placed needs to be only have half the size in the improved version. Also the particle correction step needs to be performed only once, after the convection of the level set function. When a particle passes the interface by more then its radius it is used to correct the level set function. In the current implementation the particle correction scheme is fully parallelized.

2.5 *Surface Tension*

Cohesive forces act between the molecules of a liquid. At the interface liquid-gas the molecules of the liquid phase do not have neighbors of their own phase. Since the cohesive forces of the liquid molecules are larger than that of the gas, they are attached stronger to each other on the interface than inside the uid. In order to consider the surface forces in the momentum equations, they need to be transformed into volume forces. This is done with the continuum surface force (CSF) model by Brackbill et al. [8]. The following source term $S_{CSF,i}$ needs to be added to the momentum equations:

$$S_{CSF,i} = \sigma\kappa(\phi)\partial(\phi)\frac{\partial \phi}{\partial x_i} \tag{14}$$

The surface tension coefficient σ is a material property. For the water-air interface at 20 °C it is 0.07275 N/m. The calculation of the interface curvature is straightforward with the level set method, no reconstruction is of the free surface is necessary:

$$\kappa = \nabla\left(\frac{\nabla \phi}{|\nabla \phi|}\right) \tag{15}$$

In order to activate the surface tension near the interface only, the source term is multiplied with a regularized Dirac delta function.

$$\delta(\phi) = \begin{cases} \frac{1}{2\varepsilon}\left(1 + \cos\left(\frac{\Pi\phi}{\varepsilon}\right)\right) & if\ |\phi| < \varepsilon \\ 0 & else \end{cases} \tag{16}$$

2.6 *Discretization of the Convective Terms*

For complex ow situations such as free surface ows, it is essential to employ a high order discretization method while at the same time maintain a high level of numerical stability. With that in mind the fifth-order WENO (weighted essentially non-oscillatory) scheme by Jiang and Shu [9] in the finite-difference framework is chosen. The great advantage of the WENO scheme is that it can handle large gradients right up to the shock very accurately by taking local smoothness into account. The overall WENO discretization stencil consists of three local ENO-stencils. These stencils are weighted depending on their smoothness, with the smoothest stencil contributing the most significantly. In comparison to popular high resolution schemes such as MUSCL [16] or TVD [6] schemes, the WENO scheme does not smear out the solution. Instead it maintains the sharpness of the extrema. The WENO scheme is used to treat

the convective terms for the velocities U_i, the turbulent kinetic energy k, the specific turbulent dissipation rate ω and the level set function ϕ. [3].

2.7 *Projection Method for the Pressure*

The pressure is included in the modeling procedure by employing Chorin's projection method [2] for incompressible ow. Here the actual pressure gradient is neglected in the momentum equations. Instead for each time step an intermediate velocity U_i^* is computed using the transient RANSequation:

$$\frac{\partial\left(U_i^* - U_i^n\right)}{\partial t} + U_j^n \frac{\partial U_i^n}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\nu\left(\phi^n\right)\left(\frac{\partial U_i^n}{\partial x_j} + \frac{\partial U_j^n}{\partial x_i}\right) - u_i u_j\right] + g_i + S_{CSF,i}^n \tag{17}$$

The Poisson equation for pressures is formed by calcluating the divergence of the intermediate velocity field.

$$\frac{\partial}{\partial x_i}\left(\frac{1}{\rho\left(\phi^n\right)}\frac{\partial P}{\partial x_i}\right) = -\frac{1}{\Delta t}\frac{\partial U_i^*}{\partial x_i} \tag{18}$$

The Poisson equation is solved using the Jacobi-preconditioned BiCGStab algorithm [15]. The pressure is then used to correct the velocity field, making it divergence free.

2.8 *Time Advancement Scheme*

For the time discretization a second-order accurate Adams-Bashforth scheme is used. The time step size is determined through adaptive time stepping. This ensures a stable and efficient choice of the time step. The spacial discretization is represented by the operator L. The formulation is given for the level set equation and a non-equidistant time steps.

$$\phi^{n+1} = \phi^n + \frac{\Delta t_n}{2}\left(\frac{\Delta t_n + 2\Delta t_{n-1}}{\Delta t_{n-1}} L\left(\phi^n\right) - \frac{\Delta t_n}{\Delta t_{n-1}} L\left(\phi^n\right)\right) \tag{19}$$

2.9 *The Numerical Grid*

All model equations are discretized on a Cartesian grid with a staggered arrangement of the variables. The velocity variables are defined on the center of the cell faces, while all others such as the level set function, the pressure or the variables of the turbulence model on the cell centers. This way oscillations due to velocity-pressure decoupling are avoided. At the solid boundaries of the uid domain a ghost cell immersed boundary method is employed. In this method the solution is analytically continued through the solid boundary by updating the fictitious ghost cell in the solid region by extrapolation. That way the numericaldiscretization does not need to account for the boundary conditions explicitly, instead they are enforced implicitly. The algorithm is based upon the local directional approach by Berthelsen and Faltinsen [1] which was implemented in 2D. In the present study the extrapolation scheme is decomposed into the components of the threedimensional coordinate system. The current approach has several advantages: Grid generation becomes trivial, the numerical stability and order of the overall scheme is not affected. In addition the method integrates well into the domain decomposition strategy for the parallelization of the model. Here ghost cells are used to update the values from the neighboring processors via MPI.

3 EXPERIMENT

The physical experiments of the contraction case were conducted at the laboratories of the BAW (FederalWaterways Engineering and Research Institute) in Karlsruhe, Germany. The flume is 16.50 m long and 1 m wide. The contraction is 0.5 m wide. The bed of the flume is filled with sediment particles with d_{50} = 5.5 mm. The experiments are performed with several different discharges. In the present study measurements of the runs with 150 l/s are chosen for comparison with the numerical model. With this discharge erosion occurred, but is not modeled numerically. The hydraulic situation in the flume is characterized by complex three-dimensional features of the flow. The shape of the free water surface could be described as highly turbulent and unsteady. The flume is equipped with a stationary pressure based system for the measurement of the water level elevation.

4 RESULTS

For the simulations a numerical grid with an uniform mesh width of Δh = 0:05m and 80.000 cells is used. The calculations were performed on a 8-core Intel Xeon-processor workstation. For the inflow boundary condition the discharge is fixed, but the water level is not. At the outflow the water level is specifled to be 0.315 m, with zerogradient boundary conditions for the velocities. The numerical model calculates unsteady flow and the results shown are sampled from t = 25 sec. Figure 1 shows the free surface calculated with the nu-

merical model. The black lines represent the geometric enclosure of the numerical grid. The contraction is separated into 8 pieces due to the domain decomposition strategy for the parallelization. The contour of the figure represents the water level elevation. In the contraction area a significant drop in the water level can be observed. It decreases slightly until the expansion, where it reaches its minimum. Downstream of the expansion the water level raises again. Figure 2 shows a comparison of the measured and computed water elevation along the center line of the flume. While both lines follow the same characteristics it is quite obvious that the numerical model over predicts the water level upstream of the contraction by about 2 cm. Since the outflow water level is fixed and the inflow water level is allowed to move, it means that in the numerical model the energy loss in the contraction is somewhat to high. This is probably due to the fact that the roughness was not further calibrated in the numerical model.

Figure 3 shows the free surface with the vertical axis scaled up with the factor 4. The purpose is to show how detailed the level set method resolves also local free surface structures even though the grid is rather coarse with $\Delta h = 0{:}05$m. This is partly owed to the positive effect of the particle correction algorithm on the resolution of local effects. The positive and negative particles on each side of the zero level set can be seen in Figure 4. The geometry and the coarse mesh with its high interface cells to total cells ratio is somewhat unfavorably towards the performance of the method. It costs roughly 20 % of the total computational time. A finer grid would lower that number significantly.

5 CONCLUSION

In the present paper computations of free surface flow in a long contraction are presented. The combination of particle corrected level set method and immersed boundary showed stable numerical properties. The algorithm proved to be capable to capture the free surface topology with some detail. Deviations between the numerical results and the measurements for the water level elevation on the upstream side of the contraction occured. In a further study the e_ect of the bed roughness on the water level should be examined. Comparison with experimental velocity data is underway.

ACKNOWLEDGMENTS

The author would like to thank Thorsten Hüusener and Dr. Rebekka Kopmann of the BAW in Karlsruhe for providing the data used in this paper.

REFERENCES

[1] Berthelsen, P.A., Faltinsen, O.M. 2008. A local directional ghost cell approach for incompressible viscous flow problems with irregular boundaries. Journal of Computational Physics, 227:4354-4397

[2] Chorin, A. 1968. Numerical solution of the navier stokes equations. Math. Comput., 22:745-762,

[3] Shu C.W. 1997. Essentially Non-Oscillatory and Weighted Essentially Non-Oscillatory Schemes for Hyperbolic Conservation Laws.ICASE Report

[4] Enright, D., Fedkiw,R., Ferziger, J., Mitchell, I. 2002. A hybrid particle level set method for improved interface capturing. Journal of Computational Physics, 183:83-116

[5] Harlow, F. and Welch, J. 1965. Numerical calculation of time dependent viscous incompressible flow of fluid with a free surface. Physics of Fluids, 8:2182-2189,

[6] Harten, A. 1983. High resolution schemes for hyperbolic conservation laws. Journal of Computational Physics, 49:357-393

[7] Hirt, C., Nichols, B. 1981.Volume of fluid vof method for the dynamics of free boundaries.Journal of Computational Physics, 39:201-225,

[8] Brackbill J., Kothe D., and Zemach C. A continuum method for modeling surface tension. Journal of Computational Physics, 100:335-354, 1992.

[9] Jiang, G.S., Chu, C.W. 1996. Efficient implementation of weighted eno schemes. Journal of Computational Physics, 126:202-228

[10] Osher, S., Sethian, J.A. 1988. Fronts propagating with curvature- dependent speed: Algorithms based on hamilton-jacobi formulations.Journal of Computational Physics, 79:12-49

[11] Peng, D., Merriman, B., Osher, S., Zhao, H., Kang, M. 1999. A PDE-based fast local level set method. Journal of Computational Physics, 155:410-438

[12] Schlichting, H. 1979. Boundary layer theory. McGraw-Hill Book Company

[13] Shu, C.W., Osher, S. 1988. Efficient implementation of essentially non-oscillatory shock capturing schemes. Journal of Computational Physics, 77:439-471

[14] Sussman, M., Fatemin,E., Smereka, P., Osher, S. 1994. A level set approach for computing solutions to incompressible two-phase flow. Journal of Computational Physics, 114:146-159

[15] van der Vorst, H. 1992. Bi-CGStab: A fast and smoothly converging variant of bi-cg for the solution of non-symmetric linear systems. SIAM J. Sci. Stat. Comput., 13(2):631-644

[16] van Leer B. 1979. Towards the ultimate conservative di_erence scheme V. A second order sequel to Godunovs method. Journal of Computational Physics, 32:101-136

[17] Wang, Z., Yang, J., Stern, F. 2009. An improved particle correction procedure for the particle level set method. Journal of Computational Physics, 228:5819-5837

[18] Wilcox, D.C. 1994. Turbulence modeling for CFD

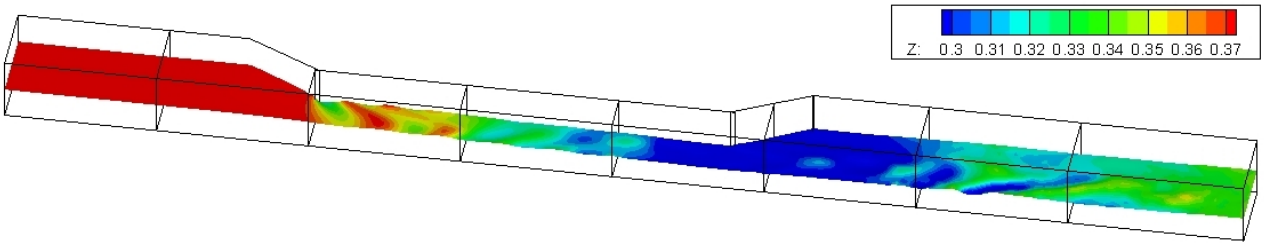

Figure 1 Computed free surface with water elevation contour

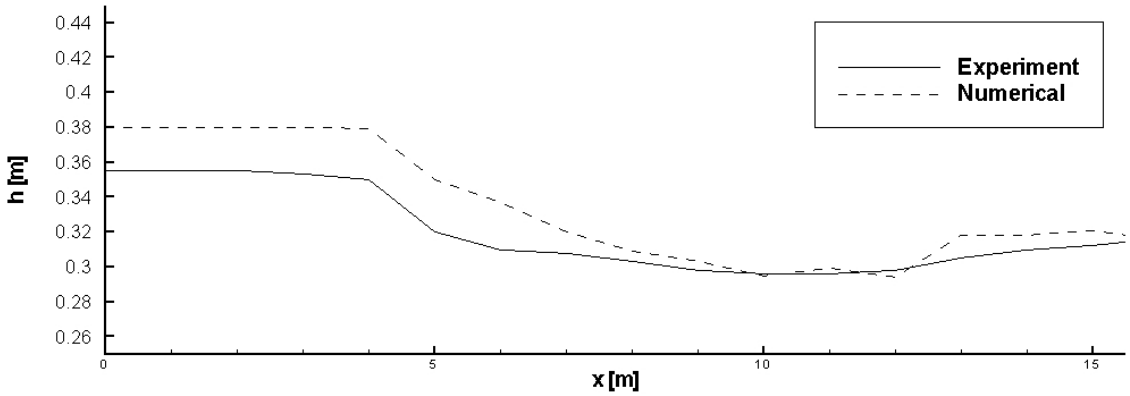

Figure 2 Water level elevation along the channel centerline

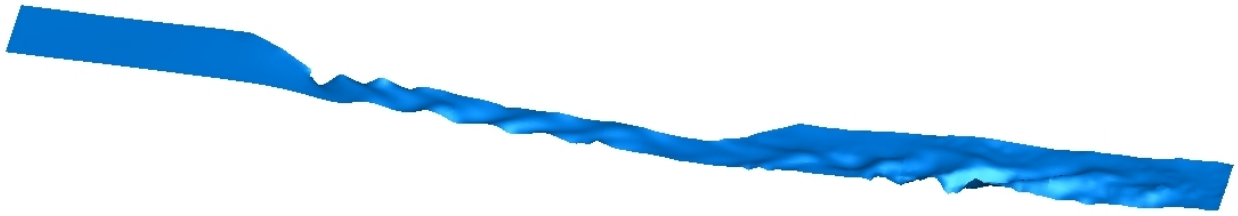

Figure 3 Computed free surface, vertical axis scaled with factor 4

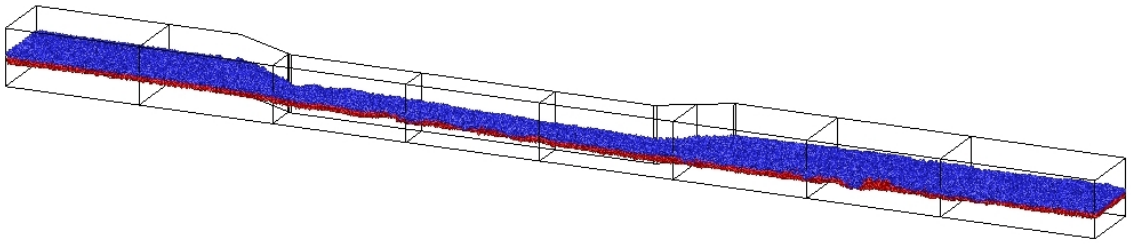

Figure 4 Negative (blue) and positive (red) particles around the interface