WCCM V
Fifth World Congress on
Computational Mechanics
July 7-12, 2002, Vienna, Austria

Dam Flooding Simulation Using Advanced CFD Methods

Mohamed Gouda*, Dr. Konrad Karner

VRVis Zentrum für Virtual Reality und Visualisierung Forschungs GmbH
A-8010 Graz, Inffeldgasse 16/II
e-mail: gouda@vrvis.at

Dr. Reinhard Tatschl

AVL List GmbH
Hans-List-Platz 1, A-8020 Graz

Key words: CFD, flooding simulation, multi-phase, finite volume, conservation equations

Abstract

Flooding, be they natural or caused by dam breaking, are a threat to mankind civilization all over the world. Today, the need for more and more water storage and electricity as a modern sources of energy in the densely populated regions are compelled to build dam along large rivers around living spaces which in turn, increases the risk of more injury to the residents. According to UN, between 1963 and 1992 floods killed more people than any other natural disaster [1].

The complexity of the environmental flows and the insufficient knowledge of the underlying physical mechanisms induce still large uncertainties in both the assessment and policies for mitigation of flood hazard.

This paper presents the VRvis flooding simulation project to develop a methodology and hydrodynamics solution based on an the AVL CFD package (SWIFT) to develop flood hazard maps.

1 Introduction

This VRvis simulation study aims on the development of a flood simulation methodology to evaluate hazard zone maps for the city areas where no history of flooding are documented. Risk analysis based on these maps determine conditions which cause flooding and define the degree of the flood risk in different regions of the city.

A simulation methodology based on Computational Fluid Dynamics (CFD) is presented here. The technique was evaluated based on hydraulics laboratory measurement and used for a real city area.

2 Flood Map Evaluation

In areas with large river basins or low land areas by the coast, floods are hazards of great significance in the case of dam failure caused by poor design or a major event such as an earthquake destroys the flood path.

The aim of the VRVis (research center of Virtual Reality and Visualization) flood simulation project is to develop a numerical solution to evaluate flood zone maps as an important component of a flood warning system. Regulatory authorities responsible for flood risk assessment and flood warning, and the insurance agents are the target requiring flood risk assessment techniques. The results from this project enable engineers to generate flood zone maps using the CFD simulation. The simulation results lead to following benefits:

- More accurate flood risk assessment for civil engineering developments; A more accurate base for calculation of financial risk.
- Improved evaluation of flood risk in a given area. More accurate flood warning systems, greater overall appreciation for the impact of floods.

3 Numerical Model

The CFD simulation of flow is a non-trivial technique when dealing with non-linear transient problems; like in the case, the fluid interfaces are unknown or moves throughout the flow, as in the case of multiphase flow processes.

The Swift simulation package is based on a Finite Volume approach and is based on general conservation equations of mass, momentum and energy, for multi-phase flows.

In the Finite Volume method, The solution domain is subdivided into a finite number of control volumes. The conservation equations are applied to each control volume units and the computational node is at the center of control volume. A direct application of Reynolds Theorem leads to the general form of the differential conservation law for an intensive property ϕ [2], [3].

We consider air and water as two separate phases during the flooding process. It is possible to calculate the volume fraction distribution for each phase by using the SWIFT multi-phase module. Conservation equations for momentum, mass and turbulence are solved for each phase.

The conservation law of mass expresses the fact that, in a fluid system, mass can not disappear from the system nor be created. The conservation of mass, or continuity is defined by:

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot \alpha_k \rho_k \mathrm{v}_k = \sum_{l=1,l\neq k}^{N} \Gamma_{kl} \qquad \text{(where } k=1,\ldots,N) \tag{1}$$

α_k is the volume fraction of phase *k*, $\mathbf{v}_k$ is the velocity phase *k*, and Γ_{kl} represents the interfacial mass exchange between phases *k* and *l*. The compatibility condition must be satisfied:

$$\sum_{k=1}^{N} \alpha_k = 1 \tag{2}$$

The continuity equation contains the unsteady term and the convection term, while the diffusion term is not present, because mass is transported only through convection [4].

The differential equation governing the conservation of momentum in a given direction for a Newtonian fluid with consideration of shear and normal stresses can be written as

$$\begin{aligned} &\frac{\partial \alpha_k \rho_k \mathrm{v}_k}{\partial t} + \nabla \cdot \alpha_k \rho_k \mathrm{v}_k \mathrm{v}_k = \\ &-\alpha_k \nabla p + \nabla \cdot \alpha_k (\boldsymbol{\tau}_k + \mathrm{T}_k^t) + \alpha_k \rho_k g + \sum_{l=1,l\neq k}^{N} \mathrm{M}_{kl} + \mathrm{v}_k \sum_{l=1,l\neq k}^{N} \Gamma_{kl} \end{aligned} \tag{3}$$

where g is the gravity vector, M_{kl} represents the momentum interfacial interaction between phases k and l, and p is pressure [5]. Pressure is assumed uniform for all phases. $\boldsymbol{\tau}_k$ is the shear stress in phase k; μ_k is molecular viscosity; T_k^t is Reynolds stress and μ_k^t turbulent viscosity.

The turbulence kinetic energy equation yields:

$$\begin{aligned} &\frac{\partial \alpha_k \rho_k k_k}{\partial t} + \nabla \cdot \alpha_k \rho_k \mathrm{v}_k k_k = \\ &\nabla \cdot \alpha_k \left(\mu_k + \frac{\mu_k^t}{\sigma_k} \right) \nabla k_k + \alpha_k P_k - \alpha_k \rho_k \varepsilon_k + \sum_{l=1,l\neq k}^{N} K_{kl} + k_k \sum_{l=1,l\neq k}^{N} \Gamma_{kl} \end{aligned} \tag{4}$$

The production term due to shear, P_k, for phase k is equal to:

$$P_k = \mathrm{T}_k^t : \nabla \mathrm{v}_k \tag{5}$$

The turbulence dissipation equation is:

$$\frac{\partial \alpha_k \rho_k \varepsilon_k}{\partial t} + \nabla \cdot \alpha_k \rho_k v_k \varepsilon_k =$$
$$\nabla \cdot \alpha_k \left(\mu_k + \frac{\mu_k^t}{\sigma_\varepsilon} \right) \nabla \varepsilon_k + \sum_{l=1, l \neq k}^{N} D_{kl} + \varepsilon_k \sum_{l=1, l \neq k}^{N} \Gamma_{kl} \tag{6}$$
$$+ \alpha_k C_1 P_k \frac{\varepsilon_k}{k_k} - \alpha_k C_2 \rho_k \frac{\varepsilon_k^2}{k_k} - \alpha_k C_4 \rho_k \varepsilon_k \nabla \cdot v_k$$

$C_{1...4}$ are the closure coefficients in the k-ε turbulence model.

4 Model Validation

Experimental data [6] obtained from a laboratory test facility for dam break case studies are compared with the SWIFT simulation results. The investigated test case comparises of combines a square shaped upstream reservoir and a 45° bend channel (see Figure 1).

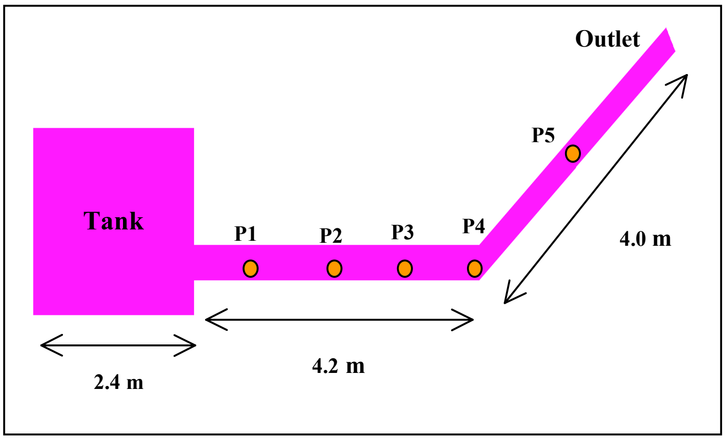

Figure 1: Plane view of the channel

The initial conditions are water at rest with the free surface 0.25 m above the bed level in the upstream reservoir and 0.01m water depth in the channel. All boundaries are solid non-slip walls expect the outlet, which is considered as static pressure.

Figure 2 shows the assumed water surface with volume fraction equal 0.5. The color scaling describes the velocity of the front. Red color shows the maximum value of the velocity and blue the minimum value.

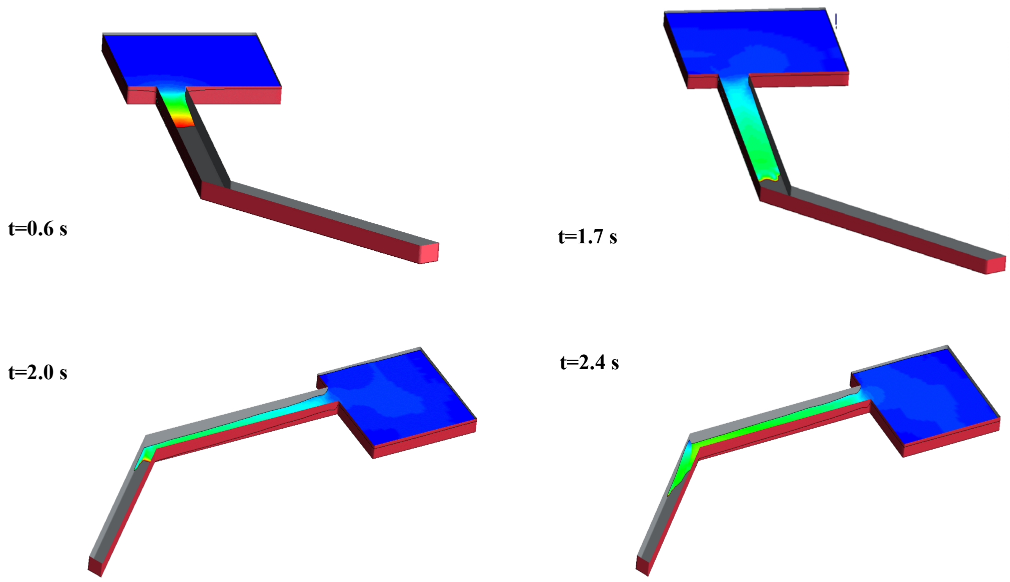

Figure 2: CFD simulation results

The water displacement front after dam breaking is of interest. Here, the displacement time in the test facility with the simulation results at defined monitor points is compared and shows good agreement between measurement and simulation (see Figure 3).

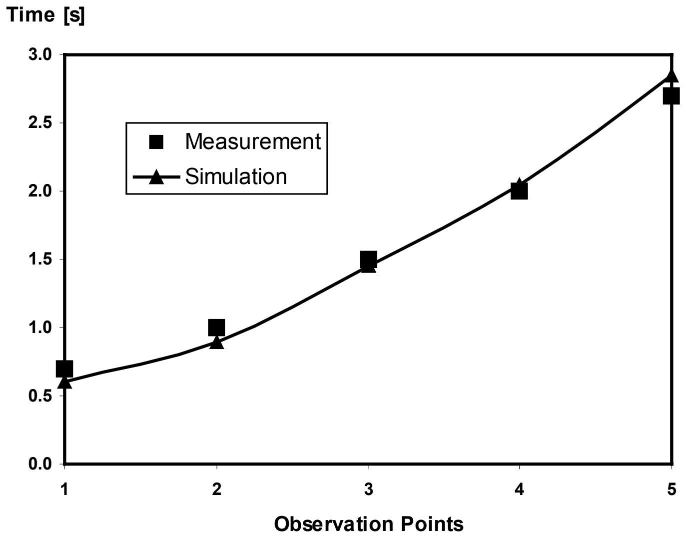

Figure 3: Comparison of displacement time between test and simulation results

5 Case Study

A real city area is the subject of this simulation study. A river that flows through center of the city, is used by several local power stations. The case of dam breaking in the upper part of the river and flooding of the center is simulated.

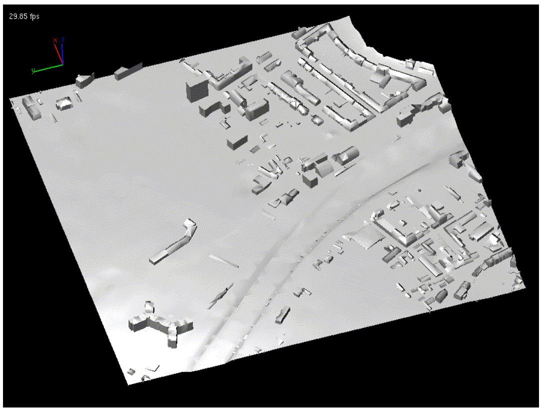

Figure 4: Simulation domain is city center area

The grid for the analyzed area was generated by using AVL FAME (Flexible Automatic Meshing Environment). It contains 225K hex elements with 2.2m in each direction. The analyzed domain is 800m x 800m and the height of mesh is extended 11m higher than the riverbed.

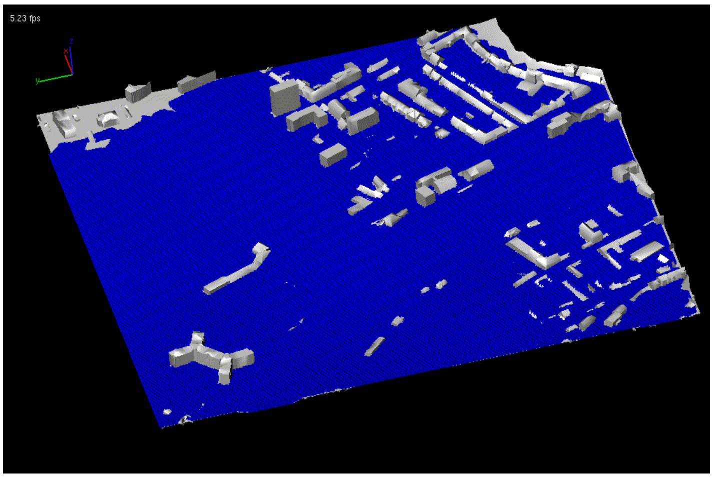

Figure 5: Grids in physical plane

The forcing condition is a discharge of 1100 m3/s, which applied suddenly by a flood wave that moves down the center of the city. The time step is 2 seconds and the total duration of simulation is 4 minutes. The total CPU time for the simulation is 82 hours on an IBM power3 with a 200MHz clock.

Figure 6 shows the degree of flooding in the different regions by volume fraction scaling. Red color shows 100% water and blue 100% air regions.

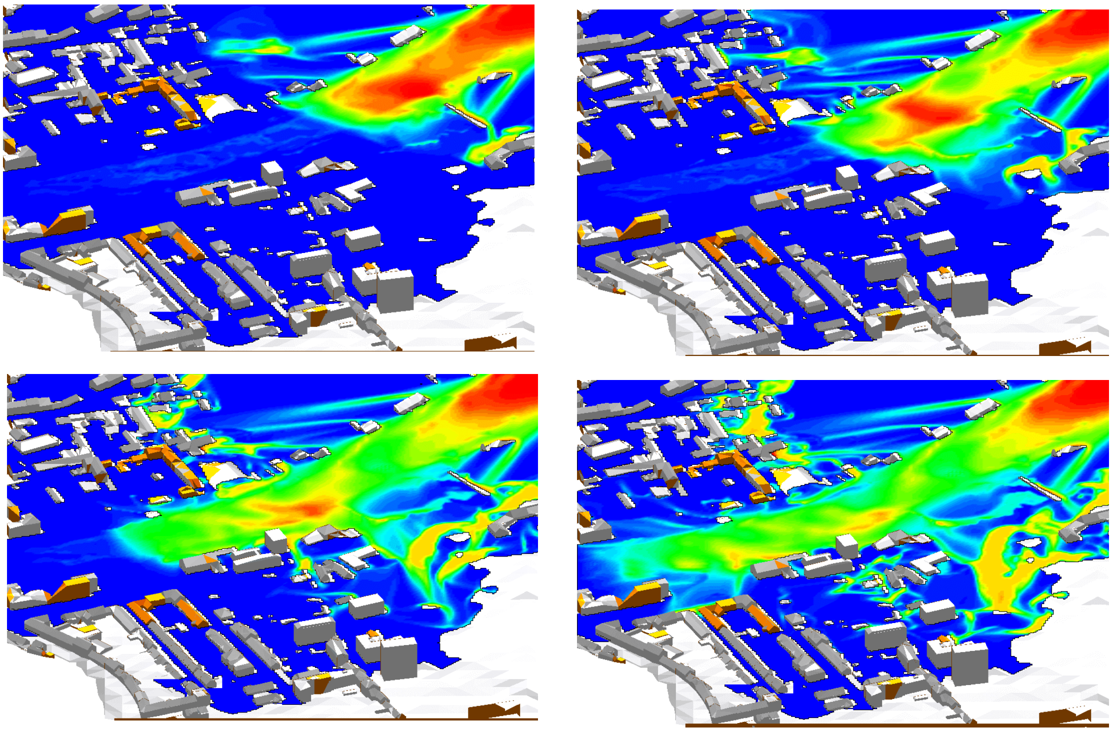

Figure 6: Computed volume fraction of the water during the flood phase

6 Conclusions

A Finite Volume algorithm based on conservation equations for multi-phase flow has been derived, discussed and applied. The high computational efficiency of this method has made it possible to provide fine details of the water circulation, velocity and pressure around the buildings in the city area during the flooding process. This simulation method enable the regulatory authorities to assess a flood zone warning in the areas where no flooding histories are documented.

References

[1] Blyth; A Telenetwork for Acquisition, Processing and Dissemination of Earth Observation Data for Monitoring and Emergency Management of Floods Hydrological Processes 11, 1359-1375, (1997).

[2] Patankar; Numerical Heat Transfer and Fluid Flow; ISBN 0-07-048740-5.

[3] B. Basara; AVL SWIFT Theory Book; AVL-List GmbH, (2001).

[4] Schiefermueller; High Order Differencing Schemes and their Application to Multi-Dimensional Flow Problems; AVL LIST GmbH, (1993).

[5] A. Alajbegovic ; AVL Multi-phase Theory Book; AVL-List GmbH (2002).

[6] P. Berufa and P. Garcia-Navarro; Two Dimensional Dam Break Flow Simulation; Int. J. Numer. Meth. Fluids; 33:35-57 (2000).