Available online at www.sciencedirect.com
ScienceDirect
Energy Procedia 82 (2015) 309 – 315
Energy Procedia

ATI 2015 - 70th Conference of the ATI Engineering Association

# Numerical Simulation of Dam-Break Problem Using an Adaptive Meshing Approach

Tommaso Fondelli[a]*, Antonio Andreini[a], Bruno Facchini[a]

[a]Department of Industrial Engineering, University of Florence, Via Santa Marta 3, Florence 50139, Italy

**Abstract**

The numerical simulation of free-surface flows is a vast topic, with applications to various fields of engineering such as aerospace, automotive, nuclear, etc. The Volume of Fluid (VOF) method represents a suitable technique to simulate free surface flows, tracking the air-liquid interface within the calculation domain. However this method requires a very fine mesh to successfully reconstruct the liquid surface, leading to very high computational costs. In this paper, VOF simulations of three-dimensional dam-break problem have been carried out using an adaptive meshing approach. Unsteady calculations have been performed exploiting the adaptive mesh feature implemented in ANSYS Fluent. In particular, a grid adaptation strategy has been defined as a way of significantly reducing the numerical effort. The main idea is to keep high resolution only locally at the air-liquid interface, minimizing numerical diffusion, and to maintain a coarse mesh size elsewhere. The dam-break problem was analyzed because it has been widely studied experimentally and numerically, representing a benchmark problem for verifying numerical models involving free-surface flows. The accuracy of the method has been assessed comparing simulation results with experimental data.


Peer-review under responsibility of the Scientific Committee of ATI 2015

Keywords: VOF; Dam-break; Adaptive mesh; Multiphase Flows; CFD.

## 1. Introduction

During the last decades increasing attention has been posed to the development of calculation methods for free-surface flows, which are involved in various fields of engineering. Free-surface flows refer to two-phase or multiphase fluid flow problems that involve two or more immiscible fluids separated by

* Corresponding author. Tel.: +39 055 2758771.
*E-mail address:* tommaso.fondelli@htc.de.unifi.it.


Peer-review under responsibility of the Scientific Committee of ATI 2015
doi:10.1016/j.egypro.2015.12.038

sharp interfaces which evolve in time. Typically, when the fluid on one side of the interface is a gas the latter is referred to as a free surface. Many methods for the treatment of the free surface are described in literature, often classified by the method for the interface treatment. An overview of the various methods available can be found in [1].

Among the computational techniques, the Volume of Fluid (VOF) method, first introduced by Hirt and Nichols [2], represents one of the most popular to study such multiphase problems. This is an interface capturing method in which the location of the interface is captured by keeping track of the volume fraction of each computational cell in the grid with respect to one of the fluid phases: cells that have a volume fraction of zero or unity do not contain an interface, whereas those that have a fractional value do. Based on this volumetric data, the free surface is reconstructed.

However, this method requires a very fine mesh to successfully reconstruct the liquid surface, leading to very high computational costs if a local grid refinement is not used. Exploiting the adaptive mesh feature implemented in ANSYS Fluent, a grid adaptation strategy has been defined as a way of significantly reducing the numerical effort. The aims of this approach are to efficiently resolve free surface, reducing numerical diffusion, as well as to maintain a coarse mesh size elsewhere.

The numerical model has been tested simulating a breaking dam flow. This issue was analysed because it has been widely studied experimentally and numerically, representing a benchmark problem for verifying numerical models involving free-surface flows. The accuracy of the method has been assessed comparing the computational results with experimental data.

## 2. Experimental set-up

The experimental test case adopted in the present work is a breaking dam flow carried out at the Maritime Research Institute of the Netherlands (MARIN) [3], sketched in Figure 1. This experiment can be seen as a simple model of green water flow on the deck of a ship. Green water can cause serious damage to deck structures, deck house, cargo and personnel. In this case an object, that for example can resemble a container, is placed on deck. The pressures that are exerted from the plunging dam are measured as function of time.

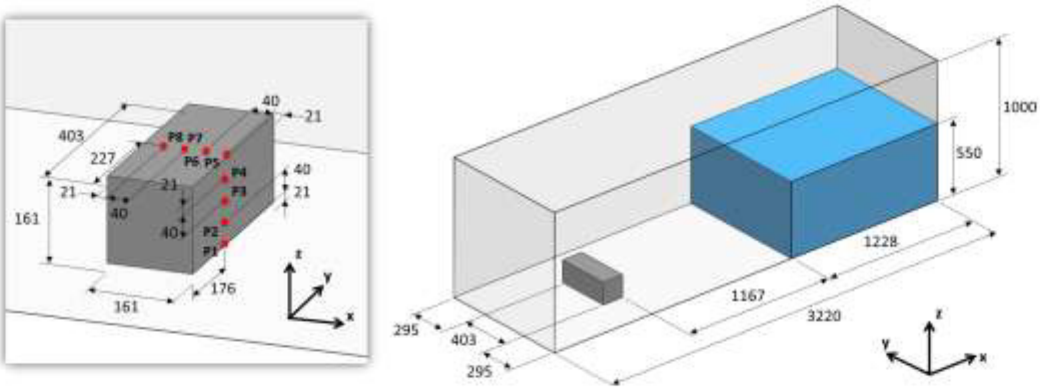


Fig. 1. Sketch of the experimental set-up (measures in mm).

The facility consisted in a 3.22 m long, 0.993 m wide and 1.0 m tall rectangular plexiglas tank having horizontal bottom. The tank is divided, by means of a separating gate, into two compartments, a 1.228 m long upstream reservoir with a water height of 0.55 m, and a downstream dry floor. The gate moved upward and could be quickly opened. A prismatic block was placed in the dry floor, 1.167 m downstream of the gate. This obstacle was 0.403 m wide, 0.161 m thick and 0.161 m tall.

In order to measure the hydrodynamic pressure acting on the obstacle in x and z directions, the block was equipped with eight pressure gauges, four on the front side and the others on the top side. The sensors position and the block orientation are illustrated in Figure 1. No information about the characteristics of pressure gauges was available in literature.

### *2.1. Test conditions*

The experiments were conducted at atmospheric conditions with water density ($\rho_w$) of 1000 kg/m$^3$ and dynamic viscosity ($\mu_w$) of $10^{-3}$ Pa s, while air density ($\rho_a$) and dynamic viscosity ($\mu_a$) were of 1.23 kg/m$^3$ and $1.73 \cdot 10^{-5}$ Pa s respectively. Under these conditions surface tension coefficient ($\sigma$) is about 0.073 Nm.

To identify the order of magnitude of the different phenomena involved, a dimensionless groups analysis has been carried out. The initial height of the water column (h) was considered as characteristic length of the problem, while the Torricelli's velocity (U) as characteristic velocity (Eq. 1), where g is the gravitational acceleration.

$$U = \sqrt{2gh} \tag{1}$$

The useful dimensionless groups for breaking dam flow are listed below:

$$Re = \rho_w hU/\mu_w = \rho_w h^3\sqrt{2g}/\mu_w = 1.81 \cdot 10^6 \tag{2}$$

$$We = \rho_w hU^2/\sigma = 2\rho_w gh^2/\sigma = 8.16 \cdot 10^4 \tag{3}$$

$$Fr = U^2/gh = \sqrt{2} \tag{4}$$

The Weber number (We) measures the relative magnitude of the inertia forces and surface tension forces, while the Froude number (Fr) the ratio of inertia and gravity forces. As shown by the dimensionless numbers, the phenomenon is driven by the inertia and gravity forces, whereas viscosity and surface tension effects are negligible.

## 3. Numerical model

The commercial code ANSYS Fluent v15.0 has been used to solve the 3D unsteady equations. One of the aims of this study is to define a numerical model able to reproduce the main features of dam breaking flow under investigation, minimizing the computational efforts. With this in mind, the flow system was treated as laminar and isothermal. Air and water were considered as incompressible fluids in according to Aureli et al. [4].

The surface tension force was modelled using the continuum surface force (CSF) model proposed by Brackbill et al. [6]. Although more accurate approaches for the surface tension forces calculation exist, like the CLSVOF approach proposed by Sussman and Puckett [7], the VOF method with CSF model has been adopted because of the problem under consideration is not a surface tension-dominant flow, according to Aureli et al. [4]. For further details about numerical setup refer to [8].

The simulations have been carried out with time step of $10^{-4}$ s, providing e a global Courant number lower than 1 for every time step and were stopped at a simulation time of 6 s. As initial configuration, the water in the reservoir is at rest. The height of the model was increased up to 1600 mm to prevent that water reaches the upper boundary during the simulation. A symmetry boundary condition has been imposed at the facility's symmetry plane (plane y); this condition leads to a considerable reduction of the computational cost. All the other model surfaces have been modelled as walls.

### *3.1. Grid Adaptation method*

As a way of significantly reducing the simulation efforts, the solution-adaptive mesh feature implemented in ANSYS Fluent has been used with the aim to confine mesh refinements at the water-air interface. One of the purposes of this approach is to efficiently resolve the liquid surface minimizing numerical diffusion of the interface.

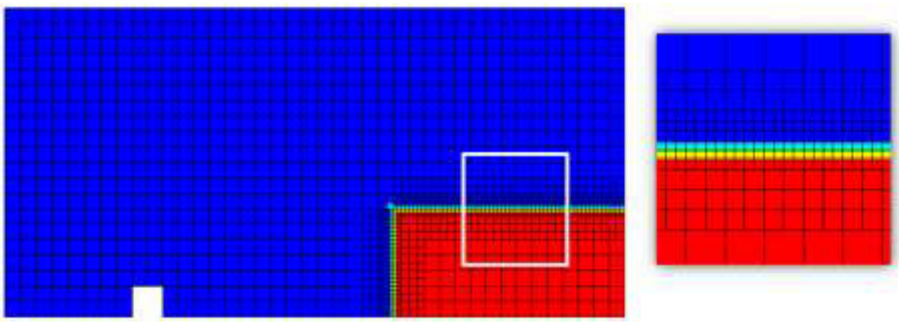

Fig. 2. Grid adaptation on the symmetry plane at initial simulation time.

An adaptation function based on the gradient of volume fraction ($\nabla\phi$) has been defined for the grid adaptation at the water-air interface. Maximum and minimum threshold values for $\nabla\phi$ were fixed before starting the simulation. The cells, containing a $\nabla\phi$ lower than the minimum threshold, are coarsened while cells having a $\nabla\phi$ higher than the maximum threshold are refined. The adaptation process is automatically and periodically executed during the unsteady calculation by means of an Execute Commands function. The Hanging Node Adaptation Process [5] is used for the hexahedral grid used in the present work: the mesh is refined by splitting “parent” cell into 8 “child cells”, adding a new point on each face of every parent cell. The Level Of Refinement (LOR), is the parameter that fixes the maximum number of the hexahedron’s splits. The mesh is coarsened by uniting the child cells to reclaim the previously parent cell. The grid cannot be coarsened any further than the original grid using the hanging node adaptation process.

Table 1. Grid dimensions and adaptation parameters adopted in the simulations.

| Run | Initial cell size (mm) | LOR | Minimum cell size (mm) | Grid size at 0 s (k nodes) | Grid size at 6 s (k nodes) |
|---|---|---|---|---|---|
| 1 | 160 | 2 | 40 | 1.1 | 16.8 |
| 2 | 160 | 3 | 20 | 1.1 | 98.9 |
| 3 | 80 | 2 | 20 | 6.8 | 92.9 |

The result of adaptation method at initial simulation time is depicted in Figure 2, where the grid in the symmetry plane is shown: the water and air are sketched in red and blue respectively.

A sensitivity analysis was performed to assess the influence of initial mesh size and the LOR parameter on the numerical predictions. The simulations that were carried out are summarized in Table 1.

## 4. Results and comparisons

A useful representation of the evolution in time of the impact is provided by the load impulse; it is defined as the integral in time of the time-varying force and quantifies the overall linear momentum absorbed by the structure starting from the initial time up to the final time [4]. The results of the sensitivity analysis are therefore compared in terms of pressure impulse.

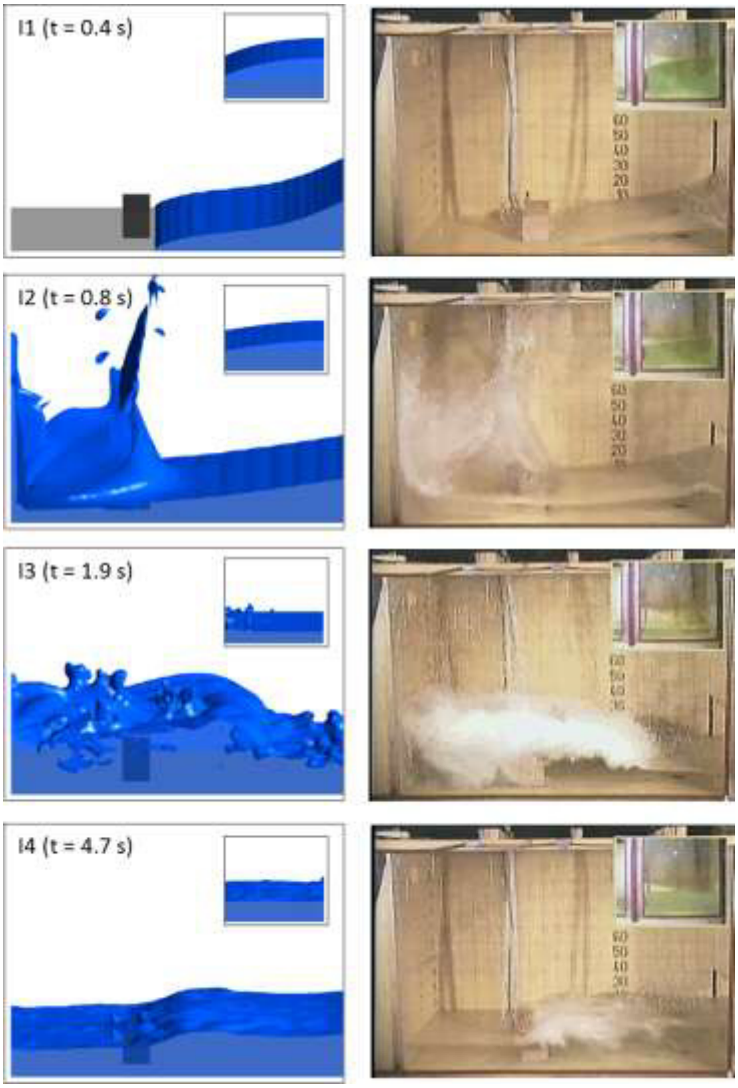


Fig. 3. Comparisons between simulation results and experimental pictures at the same flow times.

Table 2. Results sensitivity to the grid adaptation method. Integral of the pressure trends evaluated at the sensors 1 and 5. Percent deviations are referred to experimental values.

| Pressure sensor | Run-1 | Run-2 | Run-3 |
|---|---|---|---|
| P1 | 90.3 | 91.5 | 99.7 |
| P5 | 71.8 | 72.7 | 94.2 |

In detail, the integral of the pressure trends at gages P1 and P5 have been calculated both for measured and numerical data; the calculated values have been made dimensionless with respect to the experimental ones and reported in Table 2 in percentage value. The results of run-3 differ by the experimental data of 0.3% for the sensor 1 and of 5.8% for sensor 5. This simulation has been assumed as the reference, and the results have been compared with the experimental data.

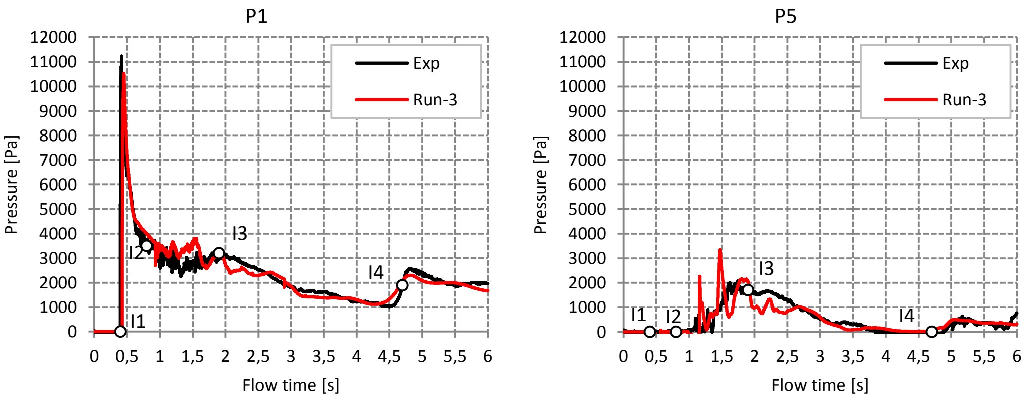


Fig. 4. Comparisons between measured and calculated pressure time series at gages P1 and P5.

In Figure 3 four snapshots of the simulation are compared with some experimental images at the same instants of time. The smaller picture at the corner of every snapshot shows the water in the reservoir. The measured and calculated pressure time series at gages P1 and P5 are sown in Figure 4. The points corresponding to the selected stages represented in the images of Figure 3 are indicated in Figure 4 over the pressure trends, in order to relate the main features of the impact pressure time history to the physical evolution of the phenomenon. There is a good agreement between simulation results and experiment.

After the gate opening, the dam-break wave spreads over the downstream area (frame I1) and hits the block approximately at time t = 0.4 s; the calculated pressure at gage P1 shows a slight delay with respect to the experimental trend. The flow is diverted from the center to the left and right sides and upward (frame I2). The sudden upward deviation of the flow produces an abrupt increase in the impact force, leading to the maximum pressure value for the sensor 1. The upward-moving jet breaks up and collapses onto the water surface of the reflecting wave creating entrapped cavities (frame I3). This produces undulations in the experimental pressure trend at flow times between 1 and 2 s, as visible in Figure 4. At this time range, the calculated results exhibit higher oscillation than the experimental data: this is essentially due to the low level of grid refinement that underestimates water breakup. Similar deviations from experimental data were also observed by other authors [9], [10].

The reflected wave moves toward the reservoir, hits the wall and is reflected back downstream, reaching the block around t = 4.7 s. This leads to the second peak on the pressure graph for sensor 1. As shown in Figure 4, for simulation times greater than 2.5 s, a very good agreement between the predicted pressure trends and the experimental data has been achieved.

## Conclusions

VOF simulations of dam-break problem have been performed using an adaptive meshing strategy, which has allowed to reduce sensibly the computational effort. The accuracy of the method was assessed comparing the results with experimental data. The numerical predictions have shown a good agreement with the experiment; in particular, the global behaviour of the liquid has been well reproduced as well as the pressure trends, especially for the sensors on the front side of the block.

## Acknowledgements

Part of this work was the subject of the MS Thesis discussed by Mr. Marco Tintori whose contributions are gratefully acknowledged.

## References

[1] Scardovelli R, Zaleski S. Direct numerical simulation of free-surface and interfacial flow. Ann. Rev. Fluid Mech.; 31, pp. 567–603, 1999.

[2] Hirt C, Nichols B. Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics; 39:201–225, 1981.

[3] Issa R, Vouleau D. 3d dambreaking. SPH European Research Interest Community SIG, Test case 2; March 2006.

[4] Aureli F, Dazzi S, Maranzoni A, Mignosa P, Vacondio R. Experimental and numerical evaluation of the force due to the impact of a dam-break wave on a structure. Advances in Water Resources; 76 (2015) 29–42.

[5] ANSYS FLUENT, Theory Guide. Release 15.0.

[6] Brackbill JU, Kothe DB, Zemach C. A continuum method for modeling surface tension. Journal of Computational Physics; 100:335–354, 1992.

[7] Sussman M, Puckett EG. A coupled level set and volume-of-fluid method for computing 3D and axisymmetric incompressible two-phase flows. Journal of Computational Physics; 162:301–337, 2000.

[8] Fondelli T, Andreini A, Da Soghe R, Cipolla L. Volume of fluid (VOF) analysis of oil-jet lubrication for high-speed spur gears using an adaptive meshing approach. Proceedings of ASME Turbo Expo 2015; GT2015-42461, JUNE 15 – 19, 2015, Montreal, Canada.

[9] Kleefsman KMT, Fekken G, Veldman AEP, Iwanowski B, Buchner B. A volume of fluid based simulation method for wave impact problems. J Comput Phys 2005; 206:363–93.

[10] Marsooli, R. and Wu, W. 3d Finite volume model of dam-break flow over uneven beds based on VOF method. Advances in Water Resources; 70:104 – 117, 2014.

**Biography**

Tommaso Fondelli is a PhD student at the Department of Industrial Engineering of Florence (DIEF). He carries out research in turbomachinery field. His main research concerns the numerical and experimental study of multiphase flows in gearbox systems for aeronautical applications.