15th Australasian Fluid Mechanics Conference
The University of Sydney, Sydney, Australia
13-17 December 2004

Simulation of The Dam Break Problem and Impact Flows Using a Navier-Stokes Solver

K. Abdolmaleki, K. P. Thiagarajan and M. T. Morris-Thomas

School of Oil and Gas Engineering
The University of Western Australia, Crawley, WA, 6009 AUSTRALIA

Abstract

The impact flow on a vertical wall resulting from a dam break problem is simulated using a Navier-Stokes (NS) solver. The NS solver uses an Eulerian finite volume method (FVM) along with a volume of fluid (VOF) scheme for phase interface capturing. The purpose of this study is to assess the accuracy of the solver for problems in the category of wave impacts. Previous experiments and other numerical solution techniques are compared with the solver's results. Different aspects of the flow such as free-surface elevation before and after the initial impact have been studied in depth. The pressure peak due to water impact on the vertical wall has also been analyzed. Water viscosity and air compressibility effects have been assessed. The significance of the time step and grid resolution are also discussed. Results show favorable agreement with experiments before water impact on the wall. However, both impact pressure and free-surface elevations after the impact depart from the experiments significantly. Hence the code is assessed to be good only for qualitative studies.

Introduction

Hydrodynamics of free-surface flows that cause impact loads on the maritime structures, has not been fully understood. The impact loads are important in designing offshore and coastal structures. Typical problems such as green water loads on ships, wave run-up on offshore structures, slam loads and sloshing loads in tanks are important in the area of naval hydrodynamics. Available theories such as potential theory, which is commonly solved by a boundary element method (BEM), cannot be successfully applied to such problems. This is because, in addition to the free-surface geometrical complexities, discontinuities in the flow and air entrainment effects in these problems cannot be treated satisfactorily by the said theories [3].

New trends are towards direct numerical solutions of NS equations. It then becomes necessary to find a proper numerical method that best serves the above mentioned complexities. In fact, the need for a computational fluid dynamics (CFD) package for naval hydrodynamics problems is highly increasing. Different numerical techniques and packages are under investigation to explore a robust methodology to numerically simulate complex free-surface and impact problems. Among the large volume of literature written on this subject, [9] provides a good review and highlights the problems associated with numerical techniques.

The dam break problem was approached mathematically by Stoker [11]. For studying green water loads on ships, this problem was investigated experimentally by Zhou et al. [13]. The dam break flow with the consequent wall impact is widely used to benchmark various numerical techniques that tend to simulate interfacial flows and impact problems (see e.g. [1, 2, 4, 5, 8, 10]). This is due to the fact that this problem includes several features of existing problems in the area of marine hydrodynamics and coastal engineering. For example, in a *shipping of water* event, the problem includes: wave run-up on the ship bow; formation of a water height above the deck (dam formation); high velocity shallow water flow (dam break); and impact of the flow on deck mounted structures (impact to the vertical wall) [2]. Therefore, the impact flow on a wall resulting from a dam break problem has scientific and practical importance.

This work uses FLUENT, a state of the art CFD package, which is widely used in both industry and academia. However, to the authors' knowledge, the software's strengths have not yet been proven in solving interfacial flow problems existing within the naval hydrodynamics framework. Therefore, the motivation of this work was to assess the software benchmark for this category of problems. The package uses a finite volume method to solve the NS equations and has several features for multi-phase flows. Among these, the VOF method is considered suitable for free-surface problems. We examine various grid sizes, time steps, air compressibility, water viscosity and turbulence effects to obtain the most realistic results from the solver. The problem dimensions are taken from [13] for comparative purposes.

The Problem Setup

A schematic of the dam break model is presented in figure 1. In this model, the tank size is $L = 3.22$ m and $H = 2.0$ m and a column of water ($L = 1.2$ m and $H = 0.6$ m) is located in the left side of the tank. For impact pressure measurements on the downstream wall, similar to the experiments in [13], a point $P(3.22\text{ m}, 0.16\text{ m})$ is defined on the wall. Free-surface elevations are recorded at stations h_1 and h_2 at distances $x_1 = 2.725$ m and $x_2 = 2.228$ m from the origin (left side wall) respectively. Notations h_1, h_2, x_1, x_2 are selected to be consistent with [4, 13].

Water is considered viscous with a constant density of $\rho_w = 998.2$ kg/m^3. The density of air is also considered constant ($\rho_a = 1.225$ kg/m^3). The flow is modeled as both laminar and turbulent, however, the plotted results are from laminar model. As shown in figure 1, the boundary conditions are all set as *wall* conditions except for the tank top, which is set as *pressure outlet*. The pressure outlet boundary condition maintains a zero gauge pressure at the defined boundary, which is desired for the tank top.

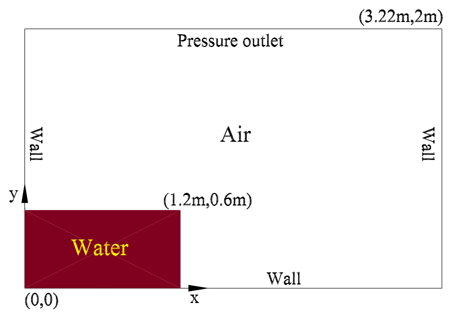

Figure 1: General layout of the dam break problem and boundary conditions.

The Numerical Setup

With the VOF method, the software allows the use of the *segregated* solver only. This solver considers the integral form of the momentum and continuity equations, which are solved sequentially. Since these equations are naturally coupled, several iterations over the solution cycle must take place to provide convergence at each time step. During the iteration process, the Poisson equation, derived from the continuity and the linearized momentum equations, is used for pressure and velocity field corrections. The convergence criterion for the continuity equation and the velocity components was selected as 0.001. For further details on the solver refer to [6].

Grid Size

The simple problem geometry allows for efficient discretisation of the domain using plane quadrilateral cells. Different grid sizes were examined to investigate the sensitivity and the accuracy of the results. The following grids were used; A1: 10mm×10mm, A2: 30mm×30mm, A3: 50mm×50mm, A4: 100mm×100mm and A5: mixture of 30mm×30mm and 15mm×15mm.

Uniform grid refining was observed to increase the computational burden as well as degrade the results. For example, for the case A1, the computation time increased dramatically and solutions did not converge at the initial water impact stage, even for $\Delta t = 10^{-9}$ s. The accuracy of both the impact pressure and the free-surface elevations depended on the grid sizes. It was also found that the results were much more sensitive to the mesh size rather than the time step. The results for fluid pressure at point P approached an upper extreme for refined grid sizes. For example, A4 under-predicts the pressure peak at P whereas A1 gives a very steep and high pressure peak at P. For local averaging of the pressure, various approximation schemes such as *Facet Average Total Pressure* were used. All the used schemes' definitions are available at [6].

Time Step

A proper time step, which provides converged results, is a function of grid size applied to the problem. For the case A5, considering $\Delta t = 0.004$ and $v_{fluid} = \sqrt{2gH} = 3.431$ m/s, the maximum value of the Courant number over the domain becomes $C = v_{fluid}\Delta t/\Delta x = 0.91$. This time step was observed to be too conservative since $\Delta t = 0.01$ s was also applied and the solutions were similar and converged. For the cases A2, A3 and A4, $\Delta t = 0.01$ s was sufficiently small to provide convergence during the entire simulation time. For the case A1, a time step of $\Delta t = 10^{-9}$ s did not provide convergence at every time step - even with 80 iterations per time step.

In the plotted results, A5 was used for spacial discretisation with a time step of $\Delta t = 0.004$ s. In A5 the grids were reduced to 15mm×15mm in the locations where the free-surface was expected. When an unsteady VOF calculation is performed in FLUENT, a time step different to the one used for the rest of the transport equations is defined for the volume fraction calculation. The time step is refined based on the input for the maximum Courant number allowed near the free-surface [6]. The Courant number for this purpose was 0.25. For the selected time steps, the solutions were all converged with the required number of iterations between 10 and 20.

Description of The Flow

Figure 2 presents some snapshots of the flow at different times. A non-dimensionalised time, $\tau = t(g/H)^{1/2}$, where H is the initial water height, is used for explaining different stages of the problem. At time $\tau = 0$ (not shown in the figure) the water column is allowed to flow. A relatively high velocity and shallow water depth flow in the x-direction quickly forms(e.g. $\tau = 2.02$). As time progresses, the flow impacts on the vertical wall at the opposite side of the tank. An upward water jet is suddenly formed that rises until gravity overcomes the upward momentum (around $\tau = 4.04$). At this moment, the jet becomes thicker and the flow starts to reverse. Due to the oncoming flow, an adverse momentum gradient is created that results in an overturning wave (around $\tau = 5.46$). This wave formation continues until the wave tip reconnects with the incident shallow water flow that now has less forward momentum. (before $\tau = 6.06$). A sudden rise in pressure occurs at the reconnection point that is of the same order of magnitude as the pressure in wall impact. This is due to the existence of high relative momentum between fluid at the wave tip and the free-surface just before the attachment. A secondary but smaller overturning wave is created due to this impact and breaks in the same manner as the first wave (around $\tau = 7.08$ and $\tau = 8.69$). At this stage, the flow has become complicated as several big and small pockets of entrained air have been created due to the first and the subsequent impacts on the free-surface. For $\tau > 8.69$, the overall momentum of the flow has reduced considerably, therefore, analysis of the flow beyond this point is of no practical significance.

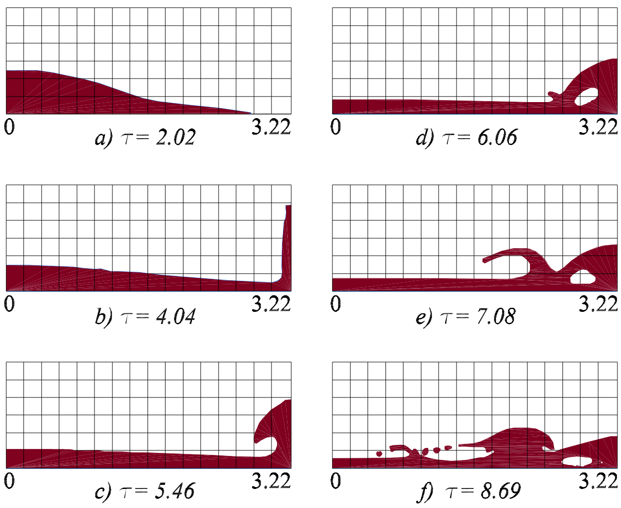

Figure 2: Dam break flow and impact against the tank wall. $\tau = t(g/H)^{1/2}$. Grids are based on case A5. The illustrated grid is of 20cm ×20cm in size and is a guide only.

Free-surface Reconstruction

In wave-structure interaction problems, free-surface location relative to an immersed body is an indicator of the impact's extent on the body. For example, in green water incidents, the key feature of the problem is to find out the initial water surface elevation just before the flow starts to run over the deck [2]. The studied problem here includes the free-surface from the smoothest geometry at the early stages of the dam break, to the most violent geometry after the water impacts on the wall.

The VOF places the free-surface where the cells are partially filled with water. Therefore, refining the mesh should provide more accurate results. During simulations, it was observed that finer grids enabled smaller entrained air pockets created in the water to be captured more effectively. However, there were two problems with mesh refinement. Firstly, it had a negative impact on the convergence of solutions and secondly, as was reported in [12], no matter how fine the mesh was defined, there was an artificially high velocity given to the air in the vicinity of the free-surface. Moreover, finer grids beyond a certain level, i.e. A5, did not improve the free-surface resolution significantly.

In a qualitative comparison, the breaking wave created after the initial impact was similar to other numerical works presented in [4, 8]. Figure 3 compares the total free-surface elevation with experimental records at stations h_1 and h_2. Results closely agree until $\tau \approx 6.5$ for both h_1 and h_2. Beyond this point, the wave travelling opposite to the main flow, reenters the free-surface and creates inaccuracies in both the experimental and the numerical results. The same scale of disagreement can be seen in the results of other numerical methods [4, 8].

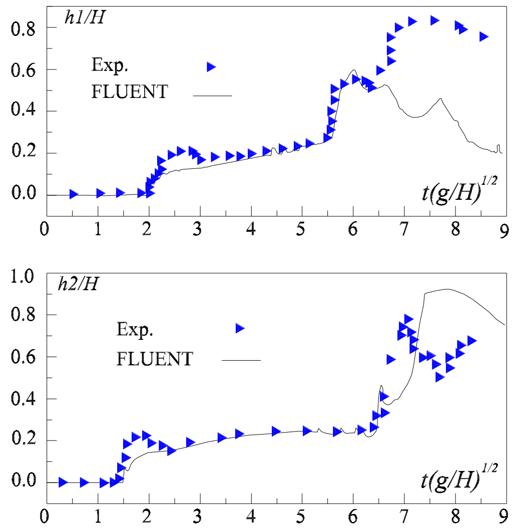

Figure 3: Total height h of the water at $x_1 = 2.725$ m (top)and $x_2 = 2.228$ m (down) from the origin. Experiment results are from [13].

A separate simulation was undertaken for studying initial stages of the dam break by tracing the water front location without existence of the downstream wall. The water column size was $L = H = 5.7$ cm. The same study was undertaken in [4] to reproduce experiments documented in [7] by different numerical techniques. These numerical techniques are the BEM, Level set and smoothed particle hydrodynamics (SPH) methods. We have added our results in figure 4 for the purpose of comparison. The figure illustrates that all of the numerical methods agree reasonably well. Also as discussed in [4], the numerical results asymptotically approach the shallow water solution as time increases. However, deviation from the experiments shows a lower progressive velocity available in the experiments. This may be due to the imperfect initial conditions in the experiments and the physical effects not considered in the numerical model. At the earliest stages of the flow i.e. $\tau < 1$, the deviation is perhaps caused by the non-uniform breaking of the diaphragm in the experiments. For $\tau > 1$, the friction on the bottom becomes an important factor in creating turbulence and delaying the progress of the water front [4].

Impact Pressure

The impact pressure at the point P can be averaged over time or location. Alternatively, it can be directly calculated (numerically only) at the point. In impact problems, as the spatial and temporal pressure gradients are high, any of the said approaches provide significantly different results. In fact, both time and local averaging underestimate the peak pressure because the peak value lasts less than $t = 0.01$ s and the surrounding points do not

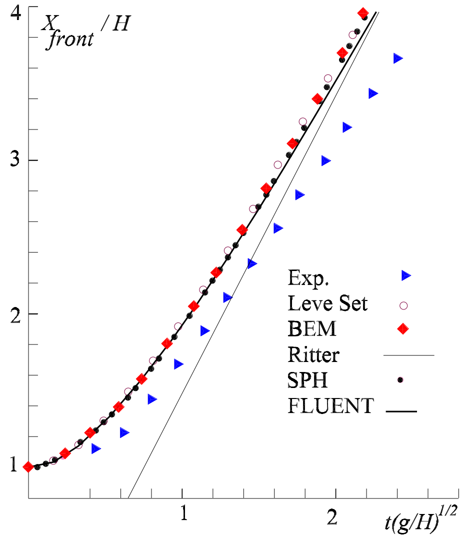

Figure 4: Time history of the water front toe evolution. $t = 0$ is the start of the flow and X_{front} is the position of the water front [4].

reach their peak value at the same time as the central point. An exact pressure measurement at P cannot be determined experimentally as pressure sensors need an area to sense the impact pressure (in [13] this area is a circle with diameter of 90 mm). However, this value can be calculated numerically. Although in this case the pressure value for a mathematical point is averaged over the neighboring nodes, improvements can be achieved by grid refinement.

In this study, the local averaging and the value of the pressure at the point P are compared. Figure 5 shows the time history of the pressure at point P. Three sets of graphs are seen in the figure. These sets are; experimental results, numerical results measured at point P and numerical results averaged over an area corresponding to the pressure sensor diameter in experiments. Different approximation schemes are used for area averaging. All results predict the two pressure peaks at about the same time instant. However, the peak magnitudes are largely different over the different schemes. At the first peak, the closest results to the experiments are the *Facet Average Total Pressure* and *Vertex Average Total Pressure* schemes whereas the other three schemes (i.e. *Facet Maximum Total Pressure, Mass-Weighted Ave. Total Pressure* and *Vertex Maximum Total Pressure*) provide results close to the pressure measured at point P (*vertex average*). The second peak is under predicted by both *Facet Average Total Pressure* and *Vertex Average Total Pressure* schemes. That is perhaps because of entrained air effects, which are not well predicted by the code. Finally, the pressure at the mathematical point P, that could be calculated numerically, was found to be much higher than the pressures measured experimentally. The existence of such a high pressure peak could not be validated by comparison with experiments.

Viscosity,Air Compressibility and Turbulence Effects

The viscosity of the water was found to be unimportant for this problem as the simulation results for viscous and inviscid options were similar. This is evidenced by [4], where although the invisid model was chosen, the numerical results compare favorably with those shown in figure 4.

Air compressibility effects were checked by assuming air as an ideal gas and assuming an adiabatic process for air compres-

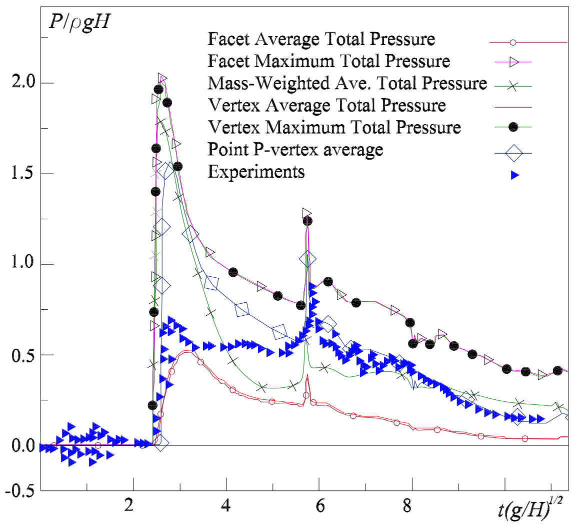

Figure 5: Impact pressures against downstream wall at point $P(3.22m, 0.16m)$. Plots are Experiment results [13], impact pressures measured at a mathematical point and averaged over an area corresponding to the pressure gauge. Different approximation schemes are plotted for area averaged case only.

sion. The software failed to simulate the whole process and unrealistic pressures appeared at the time instant of air cavity formation. This resulted in software interruption. Further studies are required to investigate the correct implementation of air compressibility as a user defined function. Finally, as was performed in [12], $K-\varepsilon$ theory was adopted to account for turbulence effects but the results were unsatisfactory and an artificial viscosity made the flow unrealistic.

Conclusion

The state of the art CFD package, FLUENT, has been employed to assess its applicability in simulating free-surface problems. In particular, we have examined the classical dam break problem and subsequent water impact on a plane vertical wall. The FLUENT results for the initial stages of the problem closely agreed with other numerical techniques and experimental results. However, there was some disagreement in water tip location between numerical results and experiments. This is perhaps due to the imperfect initial conditions and some physical effects not numerically modeled. The numerical results for the total free-surface elevation at two stations, h_1 and h_2, were in agreement while $\tau < 6.5$. The results disagree after the overturning wave reentered the free-surface i.e. $\tau > 6.5$.

The water impact pressure was numerically measured and compared with experiments. The impact pressure at point P on the wall, was measured by two methods using different averaging schemes. The averaged pressure over an area corresponding to the pressure gauge was more in line with the experimental results than the pressure measured at a mathematical point placed at the centre of the pressure gauge. Although the first peak agrees with the experimental measurements of [13], the second peak was largely underestimated. This suggests that FLUENT is acceptable for qualitative studies only.

In general, the problem after the initial impact could not be modeled with the desired accuracy. Further research is needed to strengthen the features of the software which are not suited for these types of applications. Free-surface reconstruction (complex geometry) including fluid discontinuity and the treatment of entrained air are some of the areas that require further investigations.

Acknowledgements

The first author gratefully acknowledges the funding support of the Ministry of Science, Research and Technology (MSRT) of Iran. The support of the Australian Research Council is also acknowledged.

References

[1] Brufau, P. and Garcia-Navarro, P., Two-dimensional dam break flow simulation, *Intl. J. Num. Meth. Fluids*, **33**, 2000, 35–57.

[2] Buchner, B., *Green water on ship-type offshore structures*, Ph.D. thesis, Delft University Of Technology, The Netherlands, 2002.

[3] Bulgarelli, U. P., Lugni, C. and Landrini, M., Numerical modelling of free-surface flows in ship hydrodynamics, *Intl. J. Num. Meth. Fluids*, **43**, 2003, 465–481.

[4] Colagrossi, A. and Landrini, M., Numerical simulation of interfacial flows by smoothed particle hydrodynamics, *J. Comp. Phys.*, **191**, 2003, 448–475.

[5] Greco, M., Faltinsen, O. M. and Landrini, M., Basic studies of water on deck, in *23rd Symp. on Naval Hydrodynamics*, Val de Reuil, France, 2000.

[6] Inc., F., *FLUENT 6.1 User Guide*, Fluent Inc., Centerra Resource Park 10 Cavendish Court Lebanon, NH 03766, 2003.

[7] Martin, J. C. and Moyce, W. J., An experimental study of the collapse of liquid columns on a rigid horizontal plane, *Philos. Trans. Roy. Soc. London, Ser. A*, **244**, 1952, 312–324.

[8] Nielsen, K. B., *Numerical prediction of green water loads on ships*, Ph.D. thesis, Technical University Of Denmark, Lyngby, Denmark, 2003.

[9] Scardovelli, R. and Zaleski, S., Direct numerical simulation of free-surface and interfacial flow, *Annu. Rev. Fluid Mech.*, **31**, 1999, 567–603.

[10] Shin, S. and Lee, W. I., Finite element analysis of incompressible viscous flow with moving free surface by selective volume of fluid method, *Intl. J. Num. Meth. Fluids*, **21**, 2000, 197–206.

[11] Stoker, J. J., *Water Waves: The Mathematical Theory with Applications*, Pure and Applied Mathematics, Interscience publishers, Inc., New York, 1957.

[12] Wood, D. J., Pedersen, G. K. and Jensen, A., Modelling of run-up of steep non-breaking waves, *Ocean Eng.*, **30**, 2003, 625–644.

[13] Zhou, Z. Q., Kat, J. O. D. and Buchner, B., A nonlinear 3-D approach to simulate green water dynamics on deck, Nantes, 1999, *7th Intl. Conf. Num. Ship Hydrodynamics.*