Comparison of CFD Calculations and Experiment for the Dambreak Experiment with One Flexible Wall

Rik Wemmenhove, Rune Gladsø, Bogdan Iwanowski, Marc Lefranc
Force Technology Norway AS
Sandvika, Norway

ABSTRACT

The dambreak experiment is a widely used test case for validation of CFD methods and is applied here to examine sloshing physics. Wave profiles and impact pressure profiles for the dambreak experiment are to some extent representative of those in a 2D sloshing tank. Available data from experiments show a high pressure peak for the first fluid impact. Numerical investigation of liquid dynamics and the resulting impact on the wall of the experimental chamber is presented. Interaction between fluid and structure is investigated by studying several cases with either rigid or flexible walls. Two numerical approaches are considered and their results are compared with experimental data.
The first, CFD code ComFLOW, solves Navier-Stokes equations by the Finite Volume Method and uses an improved Volume of Fluid (iVOF) method to track free surface movement. Walls of the chamber are modelled as rigid. One-phase and two-phase fluid models are applied, the latter being capable of simulating bubbles and gas entrapped in liquid.
The second approach employs Finite Elements and a well-known commercial FEM code, LS-DYNA, is used. Fluid flow modelling options include an Arbitrary Lagrangian-Eulerian scheme, which is applied here to compute liquid dynamics and impact loads on tank walls. Cases with rigid and flexible walls are examined to investigate fluid-structure interaction.

KEY WORDS: Dambreak Experiment, Sloshing, Fluid-Structure Interaction, Numerical Simulation, Wall Elasticity.

INTRODUCTION

One of the most used experiments to assess flow dynamics of confined flows is the dambreak experiment. In this experiment, a comparatively large volume of fluid is set into motion by pulling up a solid door. The fluid motion leads to a significant impact due to run-up against the wall of the tank in the experiment. A main application area of the dambreak experiment is the assessment of green water physics, see for example Kleefsman (2005).

In the present paper, the dambreak case is used to study sloshing motion onboard LNG tankers by focusing on a single impact. Full assessment of membrane type containment systems onboard LNG tankers requires thorough investigation of fluid motion and a comparison of the resulting loads with structural capacity. Realistic sloshing loads occurring in offshore situations are of stochastic nature. Long computational runs, matching the length of available experimental records, are for example presented in Iwanowski (2010).
Aside from statistical variabilities in sloshing loads, deeper investigation of the physics of one single impact deserves also attention. Kaminski and Bogaert (2009) distinguish between different types of impact and argue that wave profiles and in particular pressure profiles in a flume are representative of those in a 2D sloshing tank at low filling levels.
As part of deeper investigation of a single impact, consideration of wall elasticity is beneficial in order to gain more thorough insight into fluid motion and structural response. Kimmoun et al (2009) report significant wall deflections during their experiments where a flexible vertical wall is impacted by a breaking wave.
Fluid-structure interaction will be investigated in this paper by simulations of two programs: ComFLOW and LS-DYNA. For ComFLOW, results with single-phase (water only) and two-phase (water and air) flow are reported, while all walls are rigid. For LS-DYNA, cases with rigid and flexible vertical wall are investigated. Numerical results are compared with available experimental time series.

DAMBREAK EXPERIMENT

The experimental setup for the dambreak experiment, carried out at Maritime Research Institute Netherlands (MARIN), consists of a tank of 3.22 by 1.0 by 1.0 meter with an open roof. The right part of the tank is first closed by a door (see Figure 1). Behind the door, a 0.55 meter column of water is waiting to flow into the experimental chamber when the door is opened. This is done by releasing a weight, which almost instantaneously pulls up the door. During the experiments, measurements have been performed of water heights, pressures and forces. To measure the water height, four vertical wave probes are used, see Figure 2. Seven pressure sensors are instrumented at the left wall and are used to investigate fluid impact. Examining the experimental video, the water flows quickly towards the instrumented left wall after pulling up the door. Significant run-up against the wall is observed, followed by a back and forth motion of the water 'front', showing a gradual tendency towards a steady situation with a horizontal free surface. The focus in this paper is on the first impact against the left wall of the experimental chamber and therefore only the first 4.0 seconds of the experiment are considered.

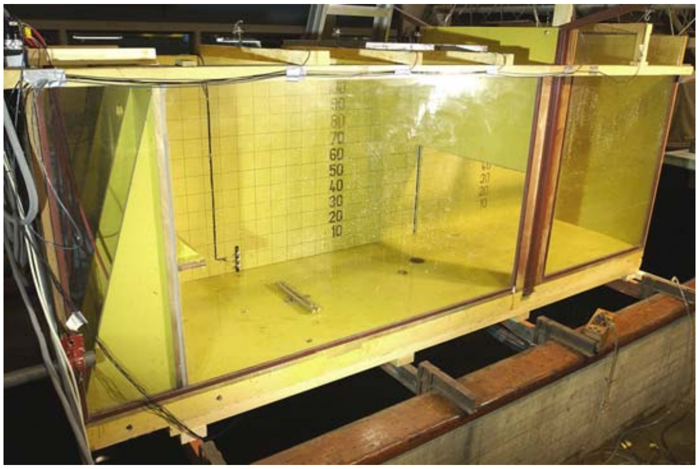

Figure 1 *Experimental setup, empty chamber*

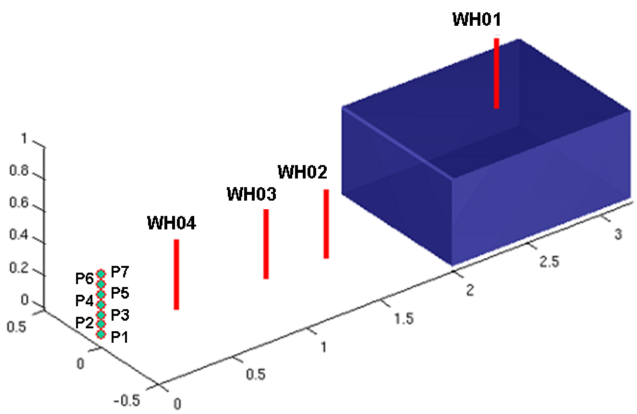

Figure 2 *Positioning of wave probes and pressure sensors*

CFD MODELLING: COMFLOW

The dambreak experiment has been one of benchmark cases for the development of the 3D Computational Fluid Dynamics solver ComFLOW. The program has been developed by University of Groningen, The Netherlands, and has been thoroughly verified against experiments during Joint Industry Projects SAFE-FLOW and ComFLOW-2. The ComFLOW code has already been applied to the sloshing problem and a comparison with large (1:10) scale experimental results was presented (Bunnik (2007), Wemmenhove (2008), Iwanowski (2009)).

The ComFLOW program can employ one of two basic physical models, either single-phase (only liquid) or two-phase (liquid and compressible gas). The latter model is recommended for complex flow problems with severe fluid impacts. A local height function is used as an improvement over the original VOF algorithm (Hirt 1981).

The theoretical model is based on one set of equations that describe behaviour of two different fluids together. The continuity and momentum equations for two-phase flow read:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \tag{1}$$

$$\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u u) + \nabla p - \nabla \cdot \left(\mu\left(\nabla u + \nabla u^T\right) - \tfrac{2}{3}\mu \nabla \cdot u\right) - \rho F = 0 \tag{2}$$

with $u = (u,v,w)$ being the fluid velocity vector, p its pressure, ρ its density, and μ its dynamic viscosity coefficient. Further, t denotes time and $F = (Fx,Fy,Fz)$ is an external body force, for example gravity. In case of two-phase flow the density ρ appears as a variable and it is necessary to close the system of equations by relating the pressure and density of the fluid. The pressure field is computed by solving the compressible form of the Poisson equation, see (Wemmenhove 2008). This equation includes an adiabatic equation of state for the compressible phase with a polytropic coefficient of γ=1.4:

$$\frac{p_{gas}}{p_{ref}} = \left(\frac{\rho_{gas}}{\rho_{ref}}\right)^{\gamma} \tag{3}$$

In the liquid phase this relation is replaced by

$$\rho = \rho_{liq} \tag{4}$$

Thus, (1) and (2) form a unified description for both phases; their only distinction being the choice for the equation of state: (3) or (4).

ComFLOW employs a Cartesian grid with a cut-cell method at boundaries of the computational domain. A staggered arrangement of variables is used, which means that pressures and densities are defined in cell centers while velocities are positioned at edges of grid cells.
Given their magnitude within the Navier-Stokes equations (1) and (2), a proper discretisation of the convective term $\nabla \bullet (\rho uu)$ deserves particular attention. The 1st order upwind differencing scheme is used in single-phase simulations. For two-phase simulations, however, this scheme leads to an excessive amount of artificial dissipation in the gaseous phase and the 2nd order upwind differencing scheme is introduced instead. This second-order scheme is used in combination with the Adams-Bashforth time integration method, while the Forward Euler time integration method is applied in single-phase simulations. More details can be found in Wemmenhove (2009).
Fluid displacement should be described properly for the complex fluid configurations observed during the dambreak experiment with water tongues, droplets and entrapped air bubbles. The free surface has to be reconstructed in a way that no additional unphysical droplets are created. A local height function is used to overcome problems of the traditional VOF algorithm with flotsam and jetsam, i.e. unphysical isolated droplets. The local height function considers a block of 3x3 (2D) or 3x3x3 (3D) cells around a free surface cell to determine the orientation of the free surface.
Some three-dimensional snapshots of fluid flow are displayed in Figure 3. Table 1 gives an overview of the computational grids employed during ComFLOW simulations. Results (water heights and pressures) will be presented for each of these computational cases.

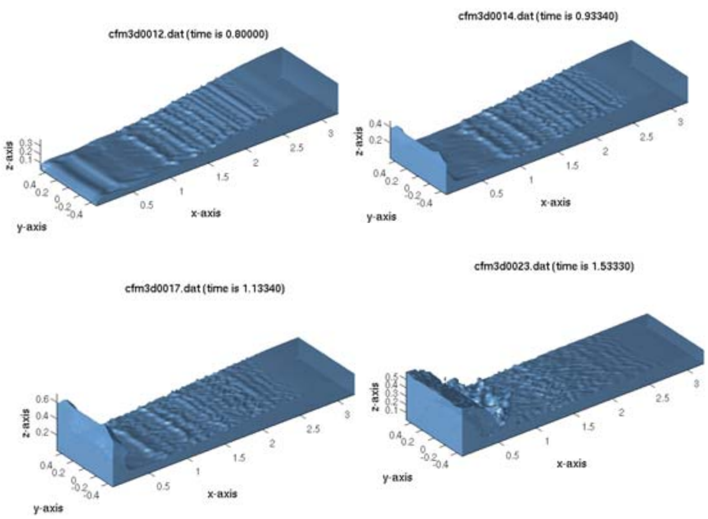

Figure 3 *Three-dimensional snapshots of ComFLOW simulation during wall impact.*

Table 1 Setup of computational cases: ComFLOW

Case	*Wall*	*dGrid*	*i*	*j*	*k*	*# cells*
C1	Rigid	0.054	59	19	17	19 057
C2	Rigid	0.027	118	38	34	152 456
C3	Rigid	0.014	236	76	68	1 219 648

FEM MODELLING: LS-DYNA

Parallel to the ComFLOW computations, fluid flow simulations have also been carried out with the Finite Element code LS-DYNA, where structural response is computed simultaneously with fluid flow.
LS-DYNA is an explicit non-linear Finite Element Method code for analysis of large deformations and dynamic response of structures. The analysis takes full account of both material and geometric non-linearities. Material non-linearities include effects of yield strength, strain hardening and failure of material. Geometric non-linearities include effects of large deformations, large rotations, membrane stretching of shell elements and local and global instability/buckling.
A 4-node Belytschko-Tsay shell element (5 degrees of freedom at each node) is used to model the walls of the experimental chamber. This type of element is very effective computationally and therefore widely used. All shell elements have a wall thickness of 20mm. In order to obtain correct stress distribution through the wall thickness, shell elements with 5 integration points through the thickness are defined.

Gravity loads are gradually applied in a quasi-static matter using a linear ramp function during the first second, and then kept constant throughout the simulation. A simplified Arbitrary Langrangian-Eulerian (ALE) formulation is applied for displacement of the interface between both fluids. This formulation consists of a Lagrangian time step followed by an advection step, where the positions of the nodes are typically moved only a small fraction of the characteristic lengths of the surrounding elements.

Table 2 Setup of computational cases: LS-DYNA

Case	*Wall*	*dGrid*	*i*	*j*	*k*	*# cells*
D1	Rigid	0.012	269	1	83	22 327
D2	Rigid	0.048	59	19	17	19 057
D3	Flexible	0.048	59	19	17	19 057

An overview of the computational grids employed in LS-DYNA is given in Table 2. The intentions of these LS-DYNA simulations are twofold.
Case D1 (with fine 12 mm grid) is meant for comparison of water heights and pressure levels with ComFLOW simulation C3. To limit computational effort this simulation has been carried out in 2D only.
Case D2 and D3 have been set up to compare the effect of wall flexibility on fluid flow in LS-DYNA. These cases have been executed on a rather coarse grid, comparable to the C1 case for ComFLOW. Modelling of wall elasticity deserves more attention, as the dynamics of the wall are highly sensitive to its positioning, wall thickness and material.

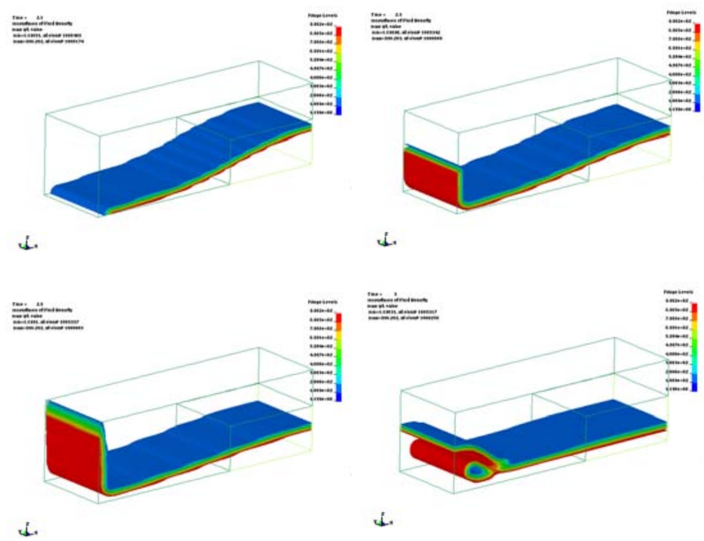

Figure 4 *Three-dimensional snapshots of LS-DYNA simulation during wall impact.*

MODELLING OF WALL ELASTICITY

The wall is positioned as in the experiment, such that it is covered by the experimental pressure transducers P1-P8 (see Figure 2). The wall is constrained with zero displacement at all edges. Effects of wall flexibility on wall dynamics and fluid flow have been examined on a relatively coarse 48mm grid in LS-DYNA. Two cases have been analyzed, the only difference being the wall material: steel to represent the rigid wall case and plexiglass to represent the flexible wall case. The (rigid or flexible) wall in LS-DYNA couples between the ALE and Lagrangian meshes. Three control points are used for each grid cell to couple ALE and Lagrangian meshes, which should minimize fluid leakage while maintaining stability in computations.

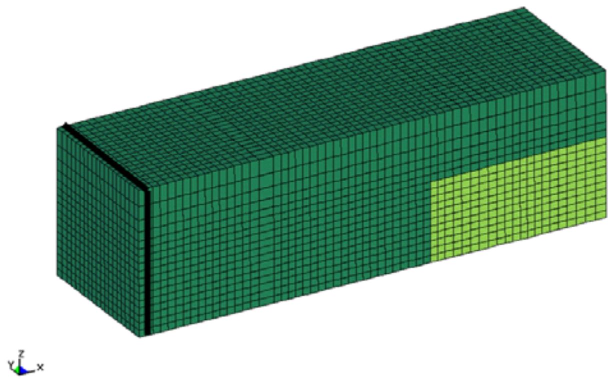

Figure 5 *Positioning of rigid/elastic wall in LS-DYNA fluid domain*

Positioning of the wall is shown in Figure 5, reference is made to the thick line in the left of the figure. There is one layer of grid cells left of the wall to give space for deflection of the flexible wall. Wall thickness has been increased to cover at least one computational cell. This has been done to minimize fluid leakage through the wall. To compensate for the enhancement of wall thickness in order to get the correct bending of the flexible wall, the elasticity modulus of the wall material has been reduced accordingly.
Steel has been applied to the rigid wall, while plexiglass has been used as material for the flexible wall. The density and stiffness of plexiglass are ρ = 1150 kgm^{-3} and E = 3.3 GPa, respectively.

RESULTS WITH COMFLOW AND LS-DYNA

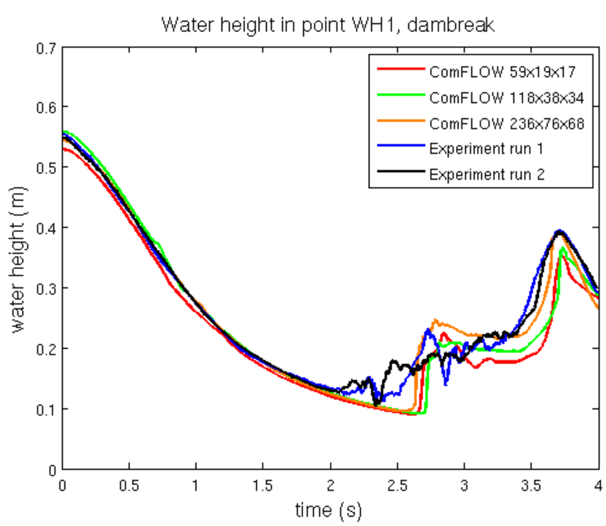

Figure 6 *Water height in reservoir, x = 2.644 m*

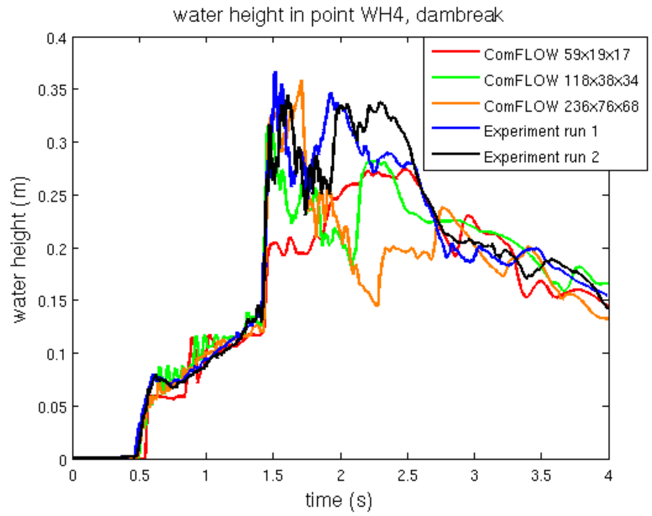

Figure 7 *Water height close to left wall, x = 0.500 m*

Water heights in ComFLOW simulations are displayed for two wave probes, WH1 and WH4 (see Figure 2 for positioning), in Figure 6 and Figure 7, respectively. The curves for the wave probe in the reservoir in Figure 6 show a good grid convergence towards experimental results. Close to the left wall, at wave probe WH4, there is a good agreement between ComFLOW water height and experimental results during the first wave impact and the following run-up. After run-up, the flow is getting much more irregular (including air entrapment), leading to increased deviations between simulations and experiments.

Pressure signals are available for both ComFLOW and LS-DYNA and results of comparison with experimental results are shown in Figure 8 and Figure 9. Pressure transducer P01 is located near the lower corner of the vertical wall and is wetted almost immediately when the wave front reaches the vertical wall. Pressure signals in Figure 8 show a rapid increase of pressure values during the first wave impact, with maximum values around 10 kPa. During run-up, just after the first impact, pressure values at P01 decrease until about 4 kPa. A second (but lower) peak in the pressure signal is visible around 1.0s after initial impact, this peak coincides in time with return of the wave front after run-up against the vertical wall. The second peak in the pressure signal is better predicted in ComFLOW simulations. Figure 9 shows the pressure development for a higher position at the vertical wall. The pressure signal at this position is mainly affected by run-up of the wave front against the vertical wall. Wave run-up is a very quick and dynamic phenomenon and earlier analyses (Wemmenhove, 2008) have shown that a fine grid resolution in ComFLOW is necessary in order to simulate this type of flow accurately. The pressure signal in LS-DYNA shows continuous oscillations for this higher vertical position, their origin is up to further investigation.

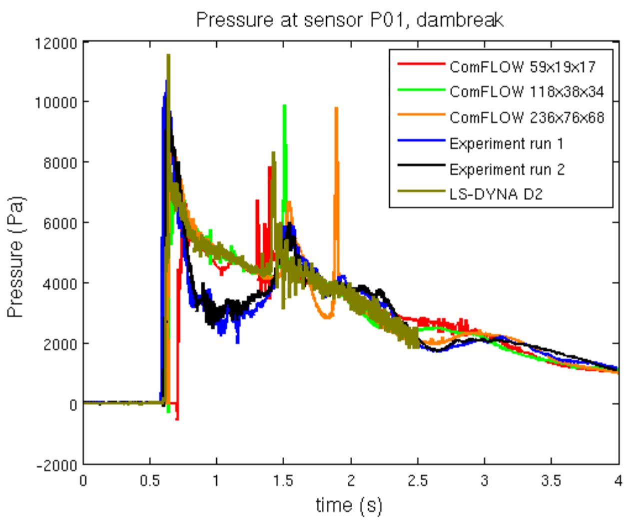

Figure 8 *Pressure at vertical wall, z = 0.030 m*

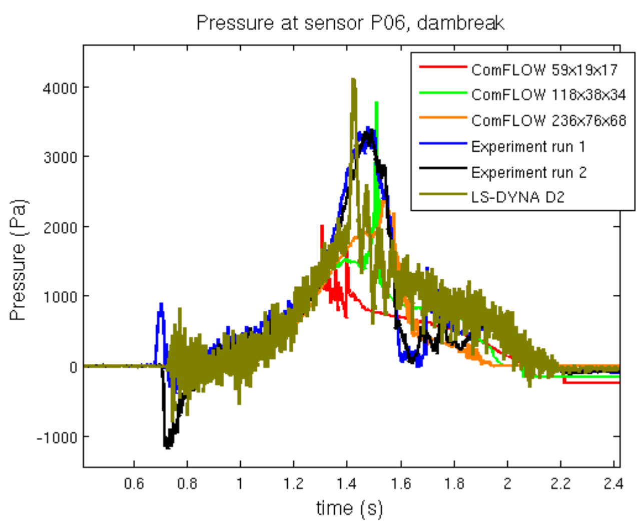

Figure 9 *Pressure at vertical wall, z = 0.405 m*

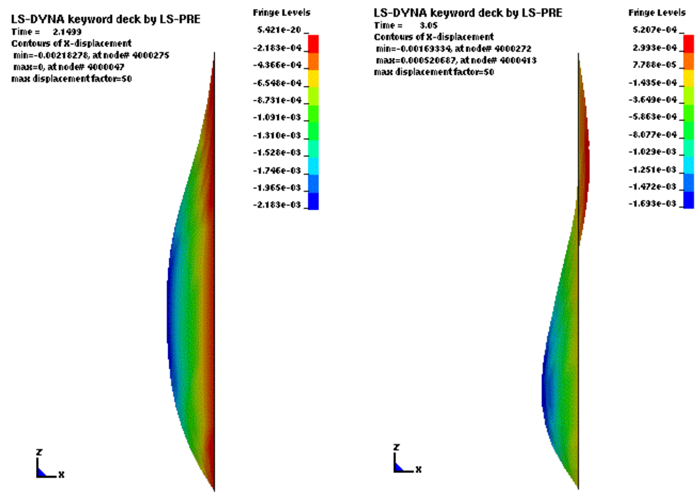

Figure 10 *Deflection of flexible wall at initial impact, t = 2.15 s (left), and after run-up against the wall, t = 3.05 s (right).*

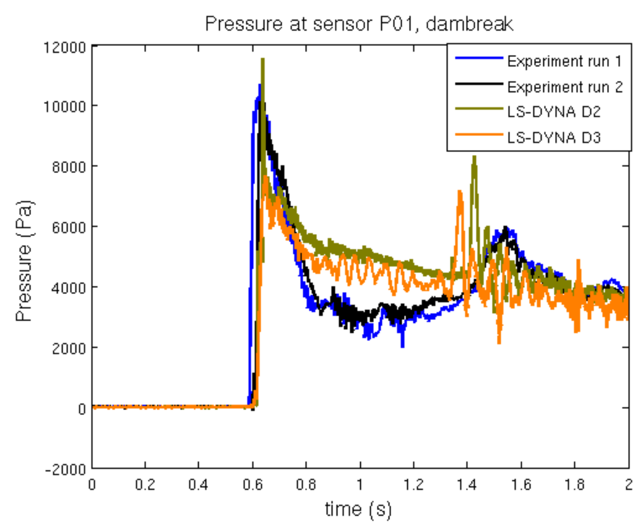

Figure 11 *Pressure at vertical wall, z = 0.030 m, for rigid and flexible wall cases in LS-DYNA.*

RESULTS WITH RIGID AND FLEXIBLE WALL

Figure 10 shows the deflection of the flexible wall at the initial fluid impact. Fluid impact occurs at the lower corner of the flexible wall, leading to a horizontal displacement up to about -5 mm (in left direction) for this lower half of the wall. Later in time, there are also positive horizontal displacements up to about +5 mm visible. The wall moves back and forth in its first vibrational mode with, after initial impact and wave run-up, excitation amplitude gradually decreasing due to damping effects.

Effects of wall flexibility on the fluid pressure signal at the wall are indicated in Figure 11. The first pressure peak is slightly lower in the simulation with flexible wall. After initial impact, the pressure signal is oscillating more for the flexible wall case. These pressure oscillations have a frequency of about 15-20 Hz during the first second after initial impact against the flexible wall.

CONCLUSIONS

The fluid flow in the dambreak experiment has been analyzed with ComFLOW and LS-DYNA simulations. The focus in this paper has been both on the fluid flow simulation itself and on the fluid-structure interaction related to a flexible wall.
Data from model experiments have been used to validate fluid flow simulations. Computational efficiency for ComFLOW simulations is higher than for LS-DYNA simulations, therefore results of LS-DYNA simulations have been reported for a 2D (but fine) grid only. Considering a fine grid, both programs give a fairly accurate prediction of water heights and pressure levels. Maximum pressure levels during

first impact against the vertical wall are up to 6-10 kPa in simulations and experiments. Pressure signals from LS-DYNA simulations oscillate more than for ComFLOW simulations and in the experimental signals. On the other hand, pressure spikes appear occasionally in ComFLOW simulations. These spikes are related to the rapid filling of previously empty grid cells within a single time step and occur for one-phase simulations only, while simulation of two-phase flow has been proven to be an effective tool to remove these spikes.

Simulations with rigid and flexible walls have shown that there is a clear effect of the fluid impact on the dynamics of the flexible wall, even when using a fairly coarse computational grid. Deflection of the plexiglass wall is up to 5 mm and the wall is subject to a vibrational motion with an excitation amplitude gradually decreasing in time. Effects of wall dynamics on the fluid flow are mainly local and fluid pressures are slightly lower and more oscillating than for the case with a rigid wall.

The current geometry with one vertical flexible vertical wall is straightforward, but more complex geometries can be considered in the near future to examine fluid-structure interaction. Choices of alternative materials and geometries open possibilities for simulation of more technical applications, such as sloshing in LNG tanks with realistic fluid flow and tank wall geometries and materials.

REFERENCES

Kimmoun, O., Malenica, S. and Scolan, Y.M. *"Fluid structure interaction occurring at a flexible vertical wall impacted by a breaking wave,"* 19th (2009) Int Offshore and Polar Eng Conf, Osaka, Japan, ISOPE, pp 308–315.

Kaminski, Miroslaw Lech and Bogaert, Hannes. *"Full Scale Sloshing Impact Tests,"* 19th (2009) Int Offshore and Polar Eng Conf, Osaka, Japan, ISOPE, pp 125–134.

Iwanowski, B., Lefranc, M. and Wemmenhove, R. *"Numerical investigation of sloshing in a tank, statistical description of experiments and CFD calculations,"* 29th (2010) Int Conf on Offshore Mech and Arctic Eng, Honolulu, USA, paper OMAE2010-20335, submitted.

Bunnik, Tim and Huijsmans, Rene. *"Large scale LNG Sloshing Model Tests,"* 17th (2007) Int Offshore and Polar Eng Conf, Lisbon, Portugal, ISOPE, pp 1893–1899.

Colagrossi, A. and Colicchio, G. *"Investigation of some naval hydrodynamics problems through a SPH methods,"* Computational Methods in Marine Engineering II (2007), Barcelona, Spain.

Hirt, C.W. and Nichols, B.D. (1981). "*Volume Of Fluid (VOF) Method for the Dynamics of Free Boundaries,*" *Journal of Computational Physics*, Vol 39, pp 201-225.

Iglesias, A.S., Rojas, L.P. and Rodriguez, R.Z. (2004). "*Simulation of anti-roll tanks and sloshing type problems with smoothed particle hydrodynamics,*" *Ocean Engineering*, Vol 31, pp 1169-1192.

Iwanowski, B., Lefranc, M. and Wemmenhove, R. *"Numerical simulation of sloshing in a tank, CFD calculations against model tests,"* 28th (2009) Int Conf on Offshore Mech and Arctic Eng, Honolulu, USA, paper OMAE2009-79051, submitted.

Jung, J., Lee, H., Park, T. and Lee, Y. *"Experimental and numerical investigation into the effects of fluid-structure interaction on the sloshing impact loads in membrane LNG carriers,"* 27th (2008) Int Conf on Offshore Mech and Arctic Eng, Estoril, Portugal, paper OMAE2008-57323.

Kim, J.W. and Kim, K. *"Response-based evaluation of design sloshing loads for membrane-type LNG carriers,"* 26th (2007) Int Conf on Offshore Mech and Arctic Eng, San Diego, USA, paper OMAE2007-29746.

Kleefsman, K.M.T., Fekken, G., Veldman, A.E.P., Buchner, B., Iwanowski, B (2005). *"A Volume-Of-Fluid Based Simulation Method For Wave Impact Problems,"* Journal of Computational Physics, Vol 31, pp 363-393

Nam, Bo-Woo and Kim, Yongwhan. *"Simulation of Two-Dimensional Sloshing Flows by SPH Method,"* 16th (2006) Int Offshore and Polar Eng Conf, San Francisco, USA.

Wemmenhove, R., Iwanowski, B., Lefranc, M., Veldman, A.E.P., Luppes, R. and Bunnik, T. *"Application of a VOF method to model compressible two-phase flow in sloshing tanks,"* 19th (2009) Int Offshore and Polar Eng Conf, Osaka, Japan, paper 2009-YHK-05.

Wemmenhove, R. (2008). "Numerical simulation of two-phase flow in offshore environments," PhD Thesis, University of Groningen, The Netherlands.

Yu, K. Chen, H.C., Kim, J.W. and Lee, Y.B. *"Numerical simulation of two-phase sloshing flow in LNG tank using finite-analytic level-set method ,"* 26th (2007) Int Conf on Offshore Mech and Arctic Eng, San Diego, USA, paper OMAE2007-29745.