MULTIPHASE 2022

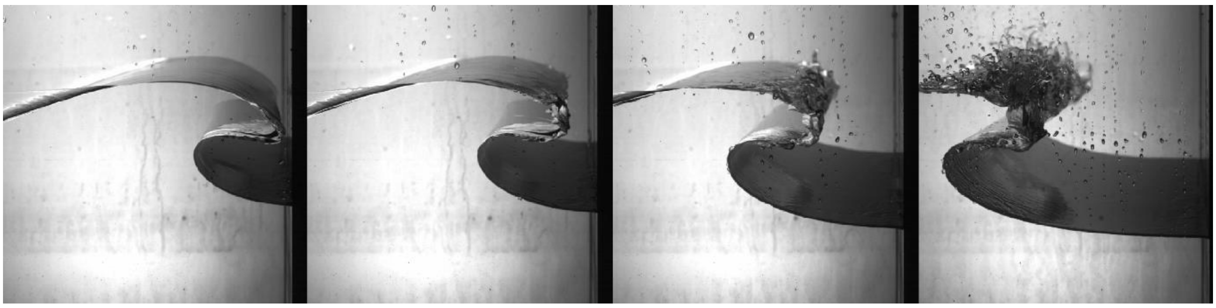
19th International workshop on trends in numerical and physical modelling for industrial multiphase flows

Jointly organised by MARIN, GTT, University of Twente and CentreBorelli ENS Paris-Saclay

Wageningen, the Netherlands-

September 21st – 23rd , 2022

Programme and abstract book



MARIN GTT Technology for a sustainable world UNIVERSITY OF TWENTE. Physics of Fluids CENTRE BORELLI

Wednesday, September 21, 2022

WELCOME AND OPENING	
11:00 – 11:15	Rodrigo Ezeta**,** *MARIN, the Netherlands*
SESSION 1:	**A NEW WAY FOR SLOSHING MODEL TESTS** Chairman: Louis Diebold *(Bureau Veritas – France)*
11:15 – 11:45	**Hybrid simulation for sloshing model tetst on hexapods** A. Durel, J. Lamaury, G. Diolex, L. Brosset, *GTT, Symétrie (France)*
SESSION 2:	**VARIOUS NUMERICAL MODELS FOR VARIOUS MULTIPHASE FLOWS** Chairman: Arthur Veldman *(University of Groningen – the Netherlands)*
11:45 – 12:15	**Modelling of a bioreactor with a multiphase CFD model** E. Gelissen, H. Krediet, *Demcon (the Netherlands)*
12:15 – 12:45	**Energy-conserving and pressure-free formulation of the two-fluid model for one-dimensional flow in ducts** J.F.H. Buist, B. Sanderse, S. Dubinkina, C.W. Oosterlee, R.A.W.M. Henkes, *CWI, TU Delft, Vrije Universiteit Amsterdam, Utrecht U., Shell Technology Centre (the Netherlands)*
12:45 – 13:45	**Lunch**
13:45 – 14:15	**Linear potential flow simulations with an open source BEM solver** M. Ancellin, *Eurobios (France)*
14:15 – 14:45	**Numerical study of change in Boil-Off Gas due to sloshing motion in cryogenic liquid tanks** Y.-D. Jung, G.-M. Jeon, J.-C. Park, *Pusan National U. (South Korea)*
SESSION 3:	**BUBBLE COLLAPSE** Chairman: Arnaud Malan *(University of Cape Town – South Africa)*
14:45 – 15:15	**Destructive dynamics of a cavitation bubble near metal surface** H. J. Sagar, B. Ould el Moctar, *U. of Duisburg-Essen (Germany)*
15:15 – 15:45	**Coffee break**
15:45 – 16:15	**Experimental and numerical investigation of thermodynamic effects on single cavitation bubble dynamics** E. Kadivar, T.-H. Phan, V.-T. Nguyen, W.-G. Park, B. Ould el Moctar, *U. of Duisburg-Essen (Germany), Pusan National U. (South Korea)*
16:15 – 16:45	**Bubble collapse regimes in the presence of a wall** D. Fuster, M. Saini, E. Tanne, S. Zaleski, M. Arrigoni, *CNRS-Sorbonne U., ENSTA Bretagne (France)*
16:45- 17:15	**Response of an equilibrium vapour bubble during liquid impact** D. van der Meer, E. Ortega Roano**,** *U. of Twente (the Netherlands)*
17:15 – 17:30	**Walk to MARIN**
17:30 – 18:30	**Presentation of The Atmosphere (ATM), the new multiphase wave lab of MARIN**

Thursday, September 22, 2022

SESSION 4:	**IMBOL PROJECT (IMPACT OF BOILING LIQUID)** Chairman: Bettar Ould el Moctar *(University of Duisburg-Essen – Germany)*
08:30 – 09:00	**Disk impact onto a liquid containing gas and vapor bubbles** E. Ortega Roano, D. van der Meer*, U. of Twente (the Netherlands)*
09:00 – 09:30	**Free surface deformation and hydrodynamic load during the impact of a corrugated disc onto water** Y. L. Fan, U. Jain, D. van der Meer*, U. of Twente (the Netherlands)*
09:30 – 10:00	**Motion and thermal pattern of an ethanol droplet deposited on heated sapphire glass** N. Kim, P. Kant, M. Souzy, D. Lohse, D. van der Meer*, U. of Twente (the Netherlands)*
10:00 – 10:30	**Impact of boiling liquids: droplets on surfaces** B. Palacios Muñiz, D. van der Meer*, U. of Twente (the Netherlands)*
10:30 – 11:00	**Coffee break**
SESSION 5:	**SLOSHING IN WINGS** Chairman: Devaraj van der Meer *(University of Twente – the Netherlands)*
11:00 – 11:30	**Experimental and numerical characterization of violent sloshing flows. A single degree of freedom approach to the problem** J. Martinez-Carrascal, L. M. González-Gutiérrez, J. Calderon-Sanchez, *U. Politécnica de Madrid (Spain)*
11:30 – 12:00	**Vertical sloshing phenomena in rectangular tanks** D. Ubaldini, D. Rossi, S. Di Giorgio, S. Pirozzoli, A. Iafrati, *Sapienza U. di Roma, CNR-INM, Istituto di Ingegneria del Mare (Italy)*
12:00 – 12:30	**Accurate CFD estimation of dynamic loads in liquid containment structures: Towards a high-fidelity simulation platform for cryogenic systems** L. Malan, B. Jones, Y. Oomar, A. Malan, S. Zaleski, *U. of Cape Town (South Africa)*
12:30 – 13:30	**Lunch**
SESSION 6:	**WAVE IMPACT TESTS IN MARIN'S FACILITIES** Chairman: Jean-Michel Ghidaglia *(Mohammed VI Polytechnic University (Morocco), ENS Paris-Saclay – France)*
13:30 – 14:00	**Influence of ullage pressure on wave impacts induced by solitary waves in a flume tank: findings from SLING project** R. Ezeta, L. Kimmoun, L. Brosset, *MARIN (the Netherlands), Ecole Centrale Marseille, GTT (France)*
14:00 – 14:30	**Wave Impact dynamics of breaking waves at phase transition conditions: findings from SLING project** R. Ezeta, L. Kimmoun, L. Brosset, *MARIN (the Netherlands), Ecole Centrale Marseille, GTT (France)*
14:30 – 15:00	**Elementary loading processes and scale effects involved in wave-in-deck type of impact loading** J. Scharnke, J. Helder, *MARIN (the Netherlands)*
15:30 – 16:00	**Bus to Arnhem**
16:00 – 19:00	**Free time in Arnhem**
19:00 – 21:30	**Gala dinner in Haarhuis Hotel**
21:30 – 23:30	**Drinks in the cellar of the Haarhuis Hotel**
23:30 – 00:00	**Bus to the hotels in Wageningen**

Friday, September 23, 2022

SESSION 7: WAVE LOADS
Chairman: Joop Helder *(MARIN – the Netherlands)*

08:30 – 09:00	**Simulation of a sloshing wave using a pressure-based multiphase compressible-incompressible flow solver** J. Muralha, L. Eça, C. M. Klaij, *IST Técnico Lisboa (Portugal), MARIN (the Netherlands)*
09:00 – 09:30	**Hydroelastic slamming in pure and aerated water** S. Tödter, J. Neugebauer, B. Ould el Moctar, *U. of Duisburg-Essen (Germany)*
09:30 – 10:00	**3D Hydroelastic impact on axisymmetric bodies** Š. Malenica, S. Seng, L. Diebold, Y.-M. Scolan, *Bureau Veritas (France)*
10:00 – 10:30	**Monitoring, mapping, tracking wave driven rocks and boulders** C. Melenaou, L. R. Diaz, D. Toal, F. Dias, *U. College Dublin, U. of Limerick (Ireland), ENS Paris-Saclay (France)*
10:30 – 11:00	**Coffee break**

SESSION 8: SIMULATION OF BREAKING WAVES
Chairman: Frédéric Dias *(University College Dublin - Ireland, ENS Paris-Saclay – France)*

11:00 – 11:30	**Numerical study of surface tension and viscosities influence on global wave shapes before impact** Z. Horriche, S. Etienne, C.Béguin, L. Brosset, *Ecole Polytechnique de Montréal (Canada), GTT (France)*
11:30 – 12:00	**Numerical modelling of variability in liquid impacts** R. A. Remmerswaal, A. E. P. Veldman, *U. of Groningen (the Netherlands)*
12:00 – 12:30	**Coherent vortical structures generated by breaking waves** S. Di Giorgio, S. Pirozzoli, A. Iafrati, *Sapienza U. di Roma, CNR-INM, Istituto di Ingegneria del Mare (Italy)*
12:30 – 13:30	**Lunch**

SESSION 9: PHOSPHATES
Chairman: Alessandro Iafrati (*Institute of Marine Engineering (Istituto di Ingegneria del Mare) – Italy)*

13:30 – 14:00	**Modeling and simulation of the multiphase pre-neutralizer reactor for NPK fertilizers production** M. Hamou, L. Khamar, S. Benjelloun, J.-M. Ghidaglia, *Mohammed VI Polytechnic U., Sultan Moulay Slimane U. (Morocco), ENS Paris-Saclay (France)*
14:00 – 14:30	**Multiscale modeling of granulation process using DEM-PBM coupling** A. Nagihi, M. Hamou, K. Ferradi, S. El Misaoui, S. Benjelloun, L. Khamar, *Mohammed VI Polytechnic U. (Morocco)*
14:30 – 15:00	**Dynamic modeling of large phosphates slurry pipeline systems in Modelica** S. Benjelloun, A. Ja, J.-M. Ghidaglia, *Mohammed VI Polytechnic U. (Morocco), Centre Borelli, ENS Paris-Saclay (France)*
15:00 – 15:30	**Assessment of RANS and LES-SGS turbulence models on the flow hydrodynamics prediction in an industrial stirred chemical reactor** S. Elmisaoui, R. Boukharfane, L. Khamar, J.-M. Ghidaglia, *Mohammed VI Polytechnic U., Mohamed V U., Sultan Moulay Slimane U. (Morocco), Centre Borelli, ENS Paris-Saclay (France)*
15:30 – 15:45	**Closing of MULTIPHASE 2022** Laurent Brosset, *GTT (France)*
15:45 – 16:15	**Coffee break**
16:15	**End of the workshop**

Hybrid simulation for sloshing model tests on hexapods

[1]*A. Durel,* [2]*J. Lamaury,* [2]*G. Diolez and* [1]*L. Brosset*
[1]Gaztransport & Technigaz (GTT), 1 route de Versailles 78470, Saint-Rémy-lès-Chevreuse, France
[2]Symétrie, 10 allée Charles Babbage 30035 Nîmes Cedex 1, France

ABSTRACT

Sloshing loads in any LNG (Liquefied Natural Gas) membrane tank on a floating structure are traditionally assessed by sloshing model tests. A small-scale model tank, instrumented with numerous dynamic pressure transducers is placed on the platform of a hexapod, a six degrees-of-freedom parallel robot. The motions imposed by the hexapod to the model tank are downscaled from calculations of the floating structure motions, taking into account the coupling with the liquid motions inside the studied tank and possibly other tanks (as for a LNG carrier). These calculations are generally made by a potential code thanks to the Boundary Element Method assuming certain simplifications such as the linearization of the free surface condition and of the coupling between the motions of the floating structure and the liquid motions within the tanks. Now, the phenomena that are to be captured during sloshing model tests are the liquid impacts against the tank walls, sometimes induced by breaking waves, which are essentially non-linear phenomena. Therefore, one can wonder how far the linear assumptions, especially on the coupling, bias the sloshing test results.

As an alternative to perform sloshing model tests on hexapods with pre-calculated forced motions, a new hybrid solution based on an External Real-Time Trajectory (ERTT) mode for the hexapods is introduced. The dynamics of the floating structure and therefore the hexapod trajectory are now both calculated in real-time. The loads induced by external waves are derived from prior seakeeping calculations. The loads induced by internal waves in the model tank are derived from the measurements of load cells placed in between the hexapod platform and the model tank.

The following operations must be performed in real-time (in between two time steps of the hexapod):

- Calculation of the hydrodynamic loads induced by the liquid on the model tank from the load cells measurement;
- Up-scaling the internal loads from model scale to full scale and low-pass filtering;
- Calculation of the full-scale floater trajectory after inclusion of the internal and external loads in the equations. The convolution of the retardation functions with the rigid body velocities is calculated for the estimation of the external loads induced by radiation;
- High-pass filtering of the floater motions to annihilate any drift;
- Change of reference system and scale (by Froude similarity) for the calculation of the hexapod platform trajectory;
- Check of the ability of the hexapod jacks to reach the new position;
- Action of the jacks to achieve the platform displacement.

Two software programs have been developed and embedded on two different real-time controllers. The first one is acquiring the load cells measurements, deriving the internal loads, calculating the floater trajectory and finally the platform trajectory. The second one is controlling the hexapod. Both programs are communicating through a direct TCP/IP connection.

Two of the four hexapods owned by *GTT* and designed by *Symétrie* have been adapted in order to enable sloshing model tests either with forced motions or in ERTT mode.

For the time being, hybrid simulation can only be used with one tank on a hexapod. Thus, the applications are limited to a LNG tank on a LNG fueled ship or on a bunker ship or for a LNG carrier in ballast condition when only one tank remains with a small quantity of LNG in order to allow keeping the tanks cold. For all these cases, the coupling between the floater motions and the liquid motions within the LNG tank is rather limited.

The tests that have been done up to now focused on the analysis of the floater motions without pressure measurements. When deactivating the load cells, the same six-degree motions are rigorously obtained by our seakeeping software and by the hexapod in ERTT mode, which validates the control mode. Tests for LNG tanks on container ships have been performed taking into account the load cells measurements. The hexapod behavior remains good. Without surprise for such an application with a small coupling, the floater motions estimated during the hybrid tests are not significantly different from those directly calculated by the potential seakeeping code.

Other applications with more important coupling are targeted in the near future. A completely different application is also envisaged for the validation of anti-roll tanks performance.

Modelling of a Bioreactor with a Multiphase CFD Model

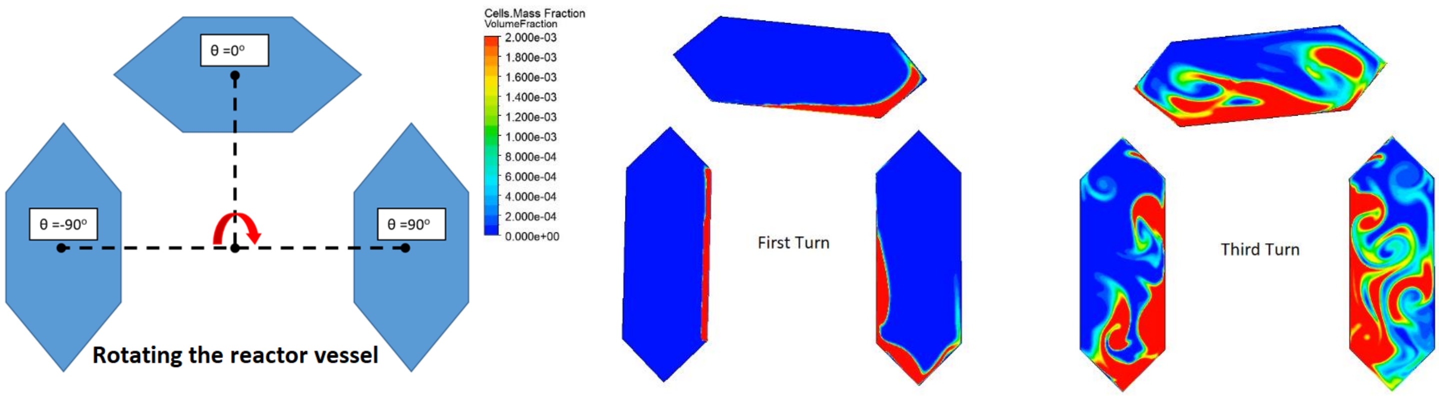
Erwin Gelissen, Harmen Krediet
Demcon Multiphysics, Kanaaldijk 29, 5683CR Best
erwin.gelissen@demcon.com

ABSTRACT

A bioreactor was developed[1] in which cells taken from a patient's body and are cultivated and after sufficient growth can be inserted back into the patient's body. This method is a form of cellular therapy which can be used against several ailments. The initial amount of donor cells is limited, so the efficiency of the cultivation process needs to be optimal. Inside the bioreactor microcarrier particles are kept in suspension in a liquid solution of nutrients. For optimal performance sufficient mixing of the microcarrier particles with nutrients is required. In particular the microcarrier particles need to be kept from settling at the bottom of the bioreactor vessel, which is achieved by periodically rotating the reactor vessel which leads to circulating flow inside the vessel.

Accurate modeling of the behavior of the suspension in the bioreactor is required in order to optimize the cultivation process. The so-called mixture model[2] was selected as an appropriate multiphase model for this system. The mixture model uses a single-fluid approach, meaning that the continuity and momentum equations are solved for the mixture. The different phases move relative to each other by using the concept of slip velocities. One phase is selected as the primary phase and transport equations are used to track the mass fractions of the secondary phases.

Simulation results give insight into the distribution/uniformity of the suspension concentration and its sensitivity to the shape of the reactor vessel, rotational velocity, vertical hold time etc.



REFERENCES

[1] SCINUS Cell Expansion System <https://www.scinus.com/scinus-cell-expansion-technology>
[2] Manninen, M., Taivassalo, V., & Kallio, S. (1996). On the mixture model for multiphase flow.

Energy-conserving and pressure-free formulation of the two-fluid model for one-dimensional flow in ducts

J.F.H. Buist[1,2], B. Sanderse[1], S. Dubinkina[3], C.W. Oosterlee[4], R.A.W.M. Henkes[2,5]
[1]Centrum Wiskunde & Informatica (CWI), [2]Delft University of Technology, [3]Vrije Universiteit Amsterdam, [4]Utrecht University,
[5]Shell Technology Centre Amsterdam

ABSTRACT

The one-dimensional two-fluid model (TFM) is a cross-sectionally averaged model for two-phase flow in pipes and channels [1], with applications in the oil and gas industry, CO_2 transport and storage, and nuclear reactor safety analysis. The model captures the main dynamics of the flow, at low computational cost. However, the model has long-standing issues with stability [2].

In this talk, we work towards an analysis of the model's nonlinear stability, by showing that the TFM (in its incompressible, isothermal form) satisfies an energy conservation equation, which arises naturally from the mass and momentum conservation equations that constitute the model. We have developed a new spatial discretization of the TFM that satisfies a discrete form of the continuous energy equation [3]. Such a discretization has been shown to prevent numerical instability and improve physical fidelity for the related model of the shallow water equations [4].

Additionally, we have developed a pressure-free version of the two-fluid model (PFTFM) [5], in which the pressure is eliminated through intricate use of the constraints of the model. This removes the pressure Poisson equation and all constraints from the model formulation, so that the discrete model can be formulated in a fully explicit manner. The cost of eliminating the pressure is that the volumetric flow rate is required as an input, for which we propose a new consistent expression based on energy conservation. Numerical experiments confirm that the PFTFM yields solutions equivalent to those of the TFM, at reduced computational cost, and with exact momentum and energy conservation.

REFERENCES

1. D. Barnea and Y. Taitel. Kelvin-Helmholtz stability criteria for stratified flow: viscous versus non-viscous (inviscid) approaches. International Journal of Multiphase Flow, 19:639–649, 1993.
2. R. W. Lyczkowski, D. Gidaspow, C. W. Solbrig, and E. D. Hughes. Characteristics and stability analyses of transient one-dimensional two-phase flow equations and their finite difference approximations. Nuclear Science and Engineering., 66:378-396, 1978.
3. J. F. H. Buist, B. Sanderse, S. Dubinkina, R. A. W. M. Henkes, and C. W. Oosterlee. Energy-conserving formulation of the two-fluid model for incompressible two-phase flow in channels and pipes. Computers & Fluids, 244:105533, 2022.
4. B. van ’t Hof and A. E. P. Veldman. Mass, momentum and energy conserving (MaMEC) discretizations on general grids for the compressible Euler and shallow water equations. Journal of Computational Physics, 231:4723–4744, 2012.
5. B. Sanderse, J. F. H. Buist, and R. A. W. M. Henkes. A novel pressure-free two-fluid model for one-dimensional incompressible multiphase flow. Journal of Computational Physics, 426:109919, 2021.

Linear potential flow simulations with an open source BEM solver

Matthieu Ancellin
Eurobios

ABSTRACT

Modeling wave-structure interactions with the linear potential flow model is a keystone of marine engineering, from shipbuilding to marine renewable energies. Many seakeeping codes implementing the Boundary Element Method (BEM) for this model are available, most of them being costly closed-source commercial codes. The development and maintenance of a free and open-source alternative can greatly benefit education and research in this field.

The open-source Python+Fortran code Capytaine [1] has been first released in 2019 and got the attention of a part of the marine energy community. Thanks to the support of the National Renewable Energy Laboratory (NREL) of the U.S. Department of Energy, its development is continuing today. Several challenges still need to be addressed to reach the level of accuracy and performance of commercial codes.

In this talk, we will present the history of the code and the theory behind the BEM numerical resolution. The roadmap for future developments of the code, as well as examples of applications to wave energy converters, will also be presented.

REFERENCES

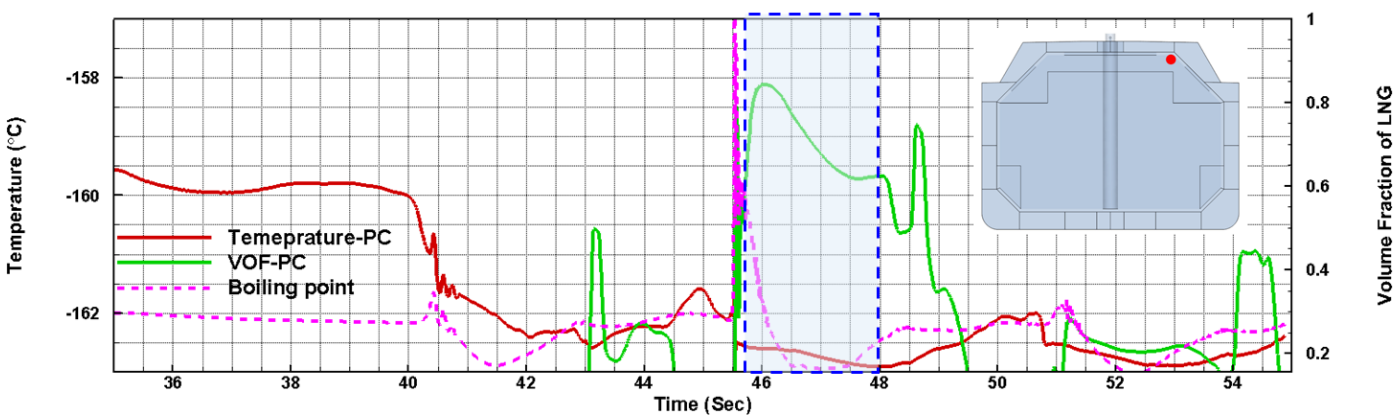
[1] Ancellin and Dias, Capytaine: a Python-based linear potential flow solver, Journal of Open Source Software, http://joss.theoj.org/papers/10.21105/joss.01341, https://github.com/mancellin/capytaine

Numerical Study of Change in Boil-Off Gas Due to Sloshing Motion in Cryogenic Liquid Tanks

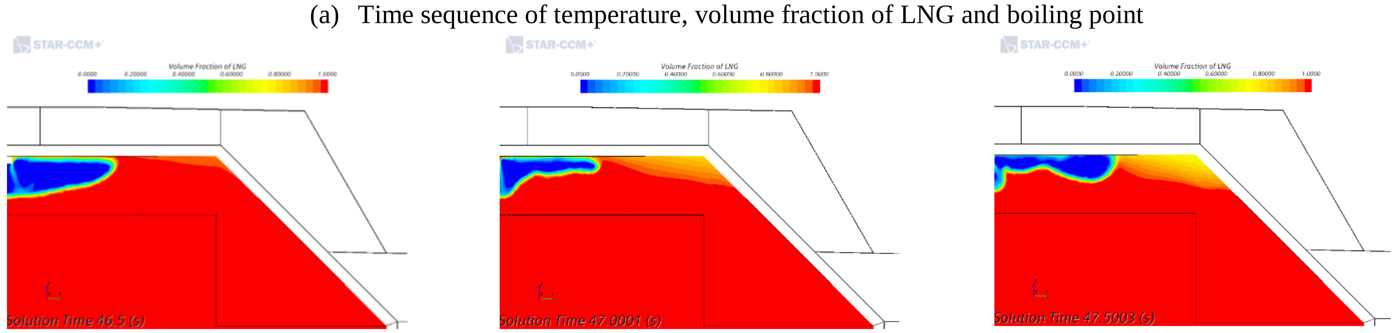
Gyu-Mok Jeon[1], Jong-Chun Park[1,]*
[1]Department of Naval Architecture and Ocean Engineering, Pusan National University, Busan, South Korea
[*]jcpark@pnu.edu

ABSTRACT

This paper mainly covers with numerical investigations on the following two issues: One is to understand the change in boil-off gas (BOG) and thermodynamic properties due to the sloshing of liquid nitrogen (LN_2) in the C-type fuel tank partially filled with cryogenic liquid by the forced roll excitation with 30 degrees. And the other is to examine the effect on BOG generation by the sloshing of the internal fluid by the sway excitation of the membrane type liquefied natural gas (LNG) storage tank. For both problems, not only the heat conduction in the insulation system but also the thermal convection due to the phase change in the fluid domain were simulated simultaneously. For the C-type tank, the temperature inside the tank and the mass flow rate of BOG were obtained numerically according to different filling rates and sloshing periods and compared with independently carried out the experimental data. As a result, it was observed that the shorter the sloshing cycle, the more the amount of BOG increased. On the other hand, in the case of a membrane-type LNG tank, as shown in Fig 1, a vaporization phenomenon associated with sloshing can be found in the upper right corner of the tank over time. Fig. 1 (a) shows the time change of temperature, volume fraction of LNG, and boiling point. At t=45.8 (s), the increase in impact pressure applied to the corner due to sloshing causes a sharp rise in the boiling point of LNG and gradually decreases until t=47.5 (s). During the subsequent t=46.0 (s) to 47.5 (s), vaporization occurs since the LNG's temperature is higher than the boiling point, and the corresponding change in a volume fraction of LNG is well illustrated in Figs 1 (b)~(d).



(a) Time sequence of temperature, volume fraction of LNG and boiling point



(b) t=46.5s (c) t=47.0s (d) t=47.5s

Figure 1 Vaporization due to sloshing in LNG tank

REFERENCES

[1] Siemens, STAR-CCM+ User Guide, Version 15.02

[2] Jeon, G. M., Park, J. C., & Choi, S. (2021). Multiphase-thermal simulation on BOG/BOR estimation due to phase change in cryogenic liquid storage tanks. Applied Thermal Engineering, 116264

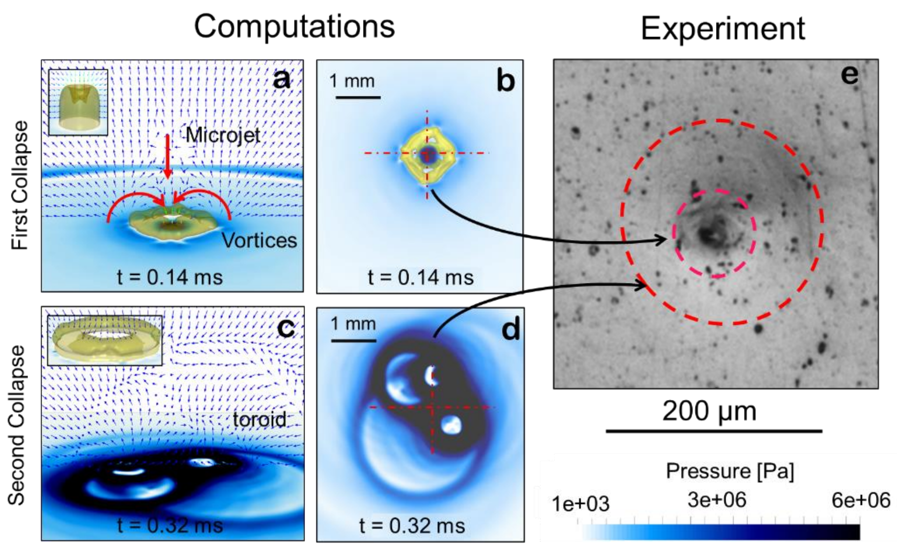
[3] Jeon, G. M., Park, J. C., Kim, J. W., Lee, Y. B., Kang, D. E., Lee, S. B., & Ryu, M.C. (2022). Experimental and Numerical Investigation of Change in Boil-off Gas and Thermodynamic Characteristics According to Filling Ratio in a C-type Cryogenic Liquid Fuel Tank. Energy, Accepted.

Destructive Dynamics of a Cavitation Bubble near Metal Surface

Hemant J. Sagar, Ould el Moctar*
Institute of Ship Technology, Ocean Engineering and Transport Systems (ISMT),
University of Duisburg-Essen, Bismarckstr. 69, 47057 Duisburg, Germany
*Corresponding author: hemant.sagar@uni-due.de

ABSTRACT

The basics of cavitation and induced damages are not very well understood in the bulk type of cavitation e.g. hydrodynamics or ultrasonic cavitation. Therefore, the experimental and numerical of a single cavitation bubble and its effects are useful in understanding the erosion mechanism as well as bubble dynamics. In the current work, the dynamics of bubbles near a flat aluminum surface were captured by a high-speed camera, and the damage on the aluminum surface was characterized by the optical microscope and profilometer. Further, flow surrounding the collapsing single cavitation bubble near a solid surface was numerically investigated considering compressibility. The three-dimensional flow was captured by the numerical code module CavitatingFOAM solves the compressible two-phase Navier-Stokes equations in an Euler-Euler approach with barotropic equations of state. In both, experiments and computations, the distance between the bubble and metal surface was changed to see the effect of distance on the dynamics of the bubble and damaging characteristics. The shapes of bubbles at various stages of collapse and rebound agreed favorably with experiments. Various flow types, pressure fields, pressure impacts on the nearby surface, and shear rate were also discussed. The correlations between simulations and experiments helped to understand the collapsing mechanism and its relations with damage patterns. A microscopic bubble collapse near the surface was also investigated to compute collapse-induced wall shear rate and flow around collapsing bubble. The results of numerical simulations qualitative quantitatively agreed well with the experimental data investigated in past with qualitative quantitatively good agreement. Overall, the current work gives broad insight into the bubble dynamics and induced damage on the aluminum surface.



REFERENCES

(1) H. J. Sagar, S. Hanke, M. Underberg, C. Feng, O. el Moctar, S. A. Kaiser, "Experimental and numerical investigation of damage on an aluminum surface by single-bubble cavitation", *ASTM- Materials Characterization and Performance (MPC),* 2018.
DOI: https://doi.org/10.1520/MPC20180038

(2) H. Sagar, B. el Moctar, "Dynamics of a cavitation bubble near a solid surface and the induced damage", *Journal of Fluids and Structures*, 92(102799), 2020.
DOI: http://doi.org/10.1016/j.jfluidstructs.2019.102799

(3) Sagar H., el Moctar B., "A Single Cavitation Bubble Collapse in Perspective of Numerical Simulations", *Numerical Towing Tank Symposium (NuTTS),* Mülheim an der Ruhr, Oct. 2021.

(4) Sagar H., el Moctar B., "A Single Cavitation Bubble Induced Damage", *41st International Conference on Ocean, Offshore & Arctic Engineering (OMAE),* Hamburg, Germany, June 2022.

Experimental and numerical investigation of thermodynamic effects on single cavitation bubble dynamics

Ebrahim Kadivar[1], Thanh-Hoang Phan[2], Van-Tu Nguyen[2], Warn-Gyu Park[2], Ould el Moctar[1]
[1]University of Duisburg-Essen, Germany
[2]Pusan National University, South Korea

ABSTRACT

The ambient temperature of the liquid may substantially affect the dynamics of a collapsing single cavitation bubble. In this study, we utilized experimental and numerical method to explore the thermodynamic effects on single cavitation bubble dynamics under various ambient temperature conditions. A series of experiments was performed to generate a single cavitation bubble at ambient temperatures between 20 and 80 °C using a laser-induced method and a high-speed. By increasing the ambient temperature, a nonspherical bubble shape with a jet flow at the bubble rebound stage was observed. Next, the numerical simulation results in terms of the bubble radius and bubble shape were validated with the corresponding experimental data. Generally, the results exhibited reasonable agreement, particularly at the later collapse and rebound stages. Critical hydrodynamic and thermodynamic mechanisms over multiple oscillation stages at different ambient temperatures were analyzed. The bubble behaviors and their intensities were numerically quantified with respect to the bubble radius, collapsing time, internal pressure, internal temperature, and phase transition rate parameters.

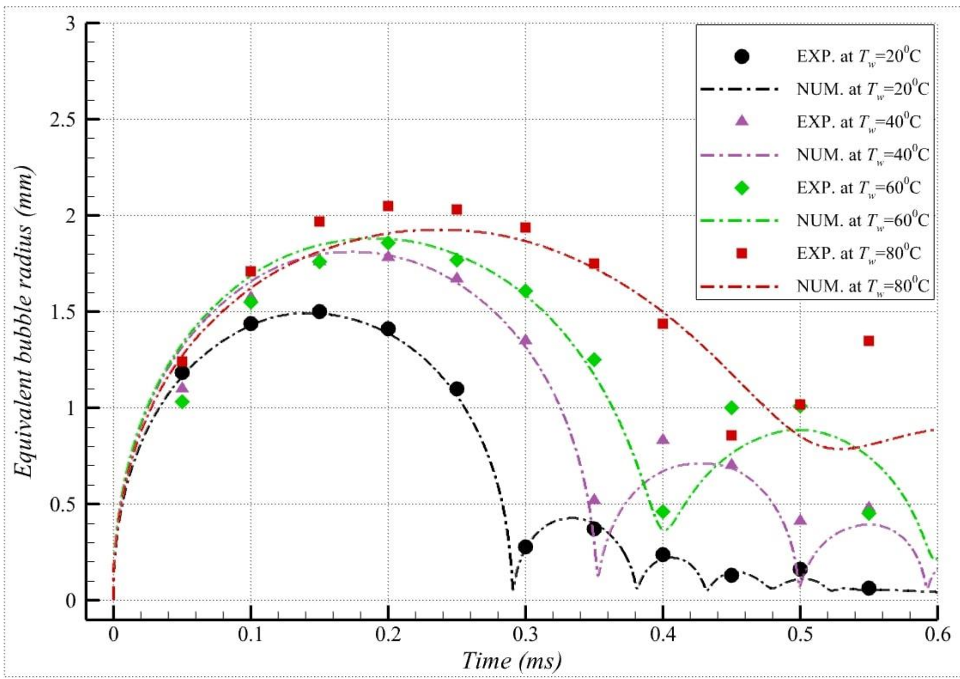


Fig. 1: Measured and computed effects on the dynamics of a collapsing singe cavitation bubble, taken from Phan et al. (2022), see reference below.

REFERENCES

Phan, T.H., Kadivar, E., Nguyen, V.T., el Moctar, O., Park, W.G. (2022). Thermodynamic Effects on Single Cavitation Bubble Dynamics under Various Ambient Temperature Conditions. Phys. of Fluids, vol. 34, 023318

Bubble collapse regimes in the presence of a wall

Daniel Fuster[(1)], Mandeep Saini[(1)], Erwan Tanne[(2)], Stephane Zaleski[(1)], Michel Arrigoni[(2)]
(1) Institut Jean Le Rond D'Alembert, CNRS-Sorbonne Université, UMR 7190, Paris.
(2) ENSTA Bertagne UMR 6027 - IRDL F-29806, Brest, France.

ABSTRACT

In this talk we will present some numerical, theoretical and experimental results that reveal the existance of different regimes of interaction between a non-spherical collapsing bubble and a wall. We show that the effective contact angle at the instant of maximum expansion is a critical parameter on determining the mechanisms of interaction between the bubble and the wall and the appearance of extremely fast jets. Using the results from the impulse theory,we will discuss the influence of the angle and liquid compressibility on the pressures generated and the phenomena observed.At first order only the pressure ratio controls the peak pressures reached inside the bubble during the collapse, the angle with the wall appearing as a second order correction. The results of Direct Numerical Simulations obtained with an all-Mach fully conservative solver [1] as well as experimental observations will be given to support the conclusions drawn from the theoretical analyses.

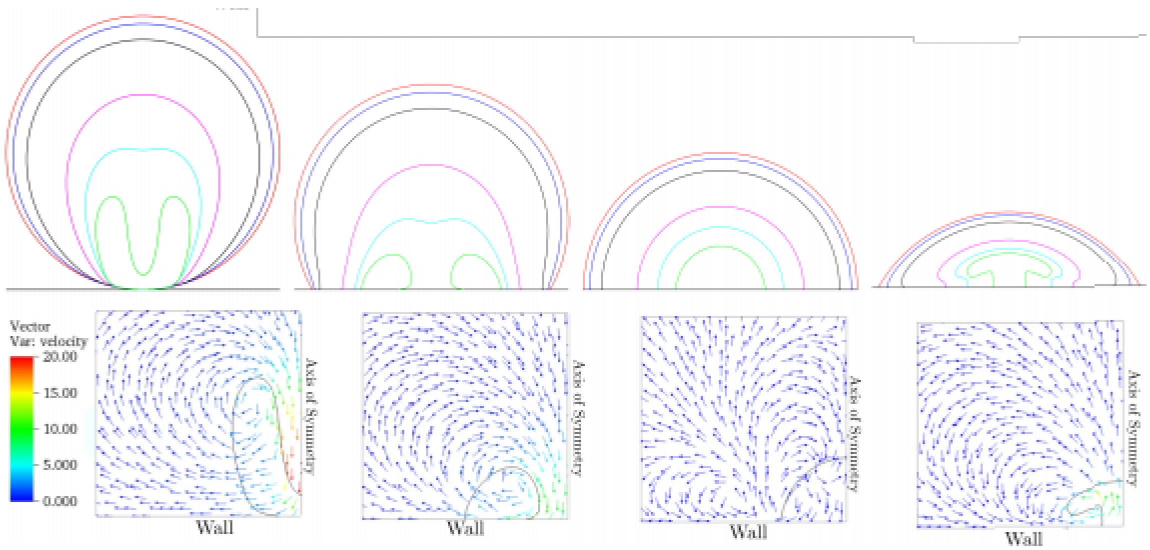


Figure 1: Figure: (Top) Bubble interface evolution of a collapsing bubbles with different initial contact angles at the instant of maximum expansion. (Bottom) Velocity field generated at the instant of minimum radius.

REFERENCES

[1] "An all-Mach method for the simulation of bubble dynamics problems in the presence of surface tension". D. Fuster, S. Popinet. Journal of Computational Physics 374, 752-768, 2018.

Response of an equilibrium vapour bubble during liquid impact

Devaraj van der Meer and Edgar Ortega Roano
University of Twente, The Netherlands

ABSTRACT

Many cryogenic liquids, such as LNG and LH2, are transported in containers where the liquid is in (or close to) thermodynamical equilibrium with its own vapour. In many such cases vapour bubbles are long lived [1] and the dynamics of these vapour pockets may play a crucial role during impact of the liquid on the container walls [2]. Here we use a theoretical and numerical approach to investigate how spherical vapour bubbles respond to external forcing, making use of the famous Rayleigh-Plesset equation for the dynamics of the bubble and the Plesset-Zwick formula for the thermal boundary layer [3]. We distinguish between a vapour-like regime, where the bubbles collapse, and gas-like behavior, where bubbles undergo damped oscillations. Subsequently we formulate conditions for the occurrence of either of the two regimes and discuss the consequence of vapour being present in clouds of small bubbles rather than in the form of a few large bubbles.

REFERENCES

[1] Andrea Prosperetti, *Vapor Bubbles*, Annu. Rev. Fluid Mech. **49**, 221–248 (2017).
[2] Frédéric Dias and Jean-Michel Ghidaglia, *Slamming: Recent Progress in the Evaluation of Impact Pressures*, Annu. Rev. Fluid Mech. **50**, 243–273 (2018).
[3] M.S. Plesset and S.A. Zwick, *A Nonsteady Heat Diffusion Problem with Spherical Symmetry*, J. Appl. Phys. **23**, 95–98 (1952);

The Atmosphere (ATM): a new facility to study wave impacts with a complete control of environmental conditions

R. Ezeta and H. Bogaert
Maritime Research Institute Netherlands (MARIN), Haagsteeg 2 6708PM, Wageningen, The Netherlands

ABSTRACT

SLING is a research Programme that was established in 2016 aiming at better understanding the physical phenomena related to wave impacts in LNG (Liquified Natural Gas) membrane tanks on floating structures and the biases induced by the use of sloshing model tests in order to assess sloshing loads. The consortium is composed of four Dutch Universities, research institutes, classification societies and industrial partners. The project was funded by NWO, the Dutch Research Council, and industrial partners. In this framework, a new facility, named *The Atmosphere* (or ATM), was designed and commissioned by MARIN [1].

Located in MARIN, Wageningen, NL, the ATM consists of a 12 m long flume tank within a cylindrical autoclave of 15 m in length and 2.5 m in diameter. Inside of the autoclave, we can independently control and monitor the ullage pressure p_u in the range $p_u \in [0.02,\ 10]$ bar, the temperature T in the range $T \in [15,\ 200]$ °C as well as the gas composition by using mixtures of either non-condensable gases of various molecular masses (Helium H_e, Nitrogen N_2, Air, Sulfur Hexafluoride SF_6) or condensable gases (water vapor). In the flume, the wave generation is driven by a piston-type wavemaker (WM). At the other end, the *impact wall* is instrumented with 100 pressure transducers of 5.5 mm in diameter. Different measurement systems provide us with the WM motion, the wave elevations at different positions along the flume and the measurement of the environmental conditions for every test. 17 windows are used for observation. Two high-speed cameras can be positioned in order to capture the wave shapes just before and during impacts.

In this talk, we present an overview of the motivation behind the commissioning of the ATM, its capabilities in the context of wave impacts, further details on its instrumentation, some highlights on tests that have been previously been performed, and some ideas for future works.

Figure 1. (top left panel) The Atmosphere (ATM) facility - (top right panel) The autoclave of the ATM - (bottom left panel) The flume inside the autoclave. At the fore front is the impact wall with its window - (bottom right panel) Three observation windows which are close to the impact wall.

Figure 1. Dedicated Instrumentations in the ATM: (top left panel) The wavemaker - (top right panel) One of the low-speed cameras - (bottom left panel) One of the HS cameras - (bottom right panel) The impact wall as seen from the inside of the flume. Here, one can see the pressure sensor array and the window that is visible in the bottom left panel of Figure 1.

REFERENCES

[1] Novakovic et al., Study of Global Wave Repeatability in the New Multiphase Wave Lab (MWL) (2020). ISOPE I 20, 3244.

Disk impact onto a liquid containing gas and vapor bubbles

Edgar Ortega Roano, Devaraj van der Meer
University of Twente

ABSTRACT

Solid-liquid impact phenomena are ubiquitous in nature and have been well studied and understood. However, the presence of gas or vapor bubbles inside the liquid may drastically change the liquid response as compressibility effects can no longer be neglected. Here, we study the impact of a disk onto a bubbly liquid using an axisymmetric boundary integral code in which gas and vapor bubbles are added to the flow. We employ simplified models for the interaction of these spherical and toroidal bubbles with the liquid, which include isothermal and adiabatic equations of state for gas bubbles, as well as the Epstein-Plesset [1] and Plesset-Zwick [2] solutions of the heat equation, which allow for heat and mass exchange in the case of vapor bubbles. We compare these models with solutions of the Rayleigh-Plesset equation for spherical bubbles in an infinite domain and, once tested, explore the influence of spherical and toroidal bubbles on the force experienced by the impacting disk.

REFERENCES

[1] Epstein, P. S., & Plesset, M. S. (1950). On the stability of gas bubbles in liquid-gas solutions. The Journal of Chemical Physics, 18(11), 1505-1509.

[2] Plesset, M. S., & Zwick, S. A. (1952). A nonsteady heat diffusion problem with spherical symmetry. *Journal of Applied Physics*, *23*(1), 95-98.

Free surface deformation and hydrodynamic load during the impact of a corrugated disc onto water

Yee Li Fan, Utkarsh Jain, and Devaraj van der Meer
Physics of Fluids Group, MESA+ Institute, Faculty of Science and Technology, University of Twente

ABSTRACT

In the literature, it has been suggested that a Kelvin-Helmholtz instability occurs on the water surface before the impact of a horizontal flat disc into the water, characterised by a dominant unstable wavelength emerging at the onset of the instability [1]. In this work, we manipulate the growth of this instability by printing axisymmetric sinusoidal disturbances of different wavelengths on the impacting discs, in order to induce a modulation of the shearing airflow between the approaching disc and the water surface and to further test this hypothesis. From the experimental results, a shearing airflow with a small wavelength was found to suppress the dominant unstable wavelength on the water surface. Further increasing the imposed wavelength on the discs forced the wavelength on the water surface to shift towards the imposed wavelength, whereas a resonant response was observed when the imposed wavelength approaches the theoretical dominant unstable wavelength. The modulated shearing airflow is thus shown to alter the cavity dynamics, which in turn may influence the hydrodynamic load experienced by the disc, which is measured for a λ = 17 mm corrugated disc by measuring the local pressures at different locations. Finally, the mechanism for the controlled growth of instability and the impact pressures on a corrugated disc are discussed in comparison to a flat impactor.

REFERENCES

1. U. Jain, A. Gauthier, D. Lohse & D. van der Meer, Air-cushioning effect and Kelvin-Helmholtz instability before the slamming of a disk on water, Physics Review Fluids, 6(4), L042001 (2021).

Motion and thermal pattern of an ethanol droplet deposited on heated sapphire glass

Nayoung Kim, Pallav Kant, Mathieu Souzy, Detlef Lohse, and Devaraj van der Meer
Physics of Fluids Group, Max Planck Center Twente for Complex Fluid Dynamics, University of Twente, 7500 AE Enschede, The Netherlands

ABSTRACT

When the droplet is deposited on the solid surface, the evaporating rate near the edge is higher than the other parts and radial convection toward the edge of the droplet is induced to replenish it [1]. A previous study [2] highlighted that the direction of flow depends on the ratio between thermal conductivities of the liquid and substrate; the high thermal conductivity of the solid surface makes the droplet have a higher temperature near the edge and consequent lower surface tension. This surface tension gradient along the droplet interface produces Marangoni flow from the edge to the apex.

In this study, we show that Marangoni flow can withhold the droplet from spreading on the substrate and may generate spontaneous movement of the droplet. By depositing the ethanol droplet on heated sapphire glass (temperature of the substrate, T_s: 45-70°C), we saw that the maximum radius of the droplet decreases with increasing T_s as a result of thermal Marangoni flow. In other words, as T_s increases, the droplet contracts more and has a larger apparent contact angle. The droplet motion is driven by an asymmetry of contact angle in the droplet azimuthal direction, and it moves faster in higher T_s cases. We confirmed that the contact angle remains constant to the end of evaporation, which propels the droplet throughout its lifetime.

We captured the thermal pattern on the droplet interface using an infrared camera [Fig. 1(a)]. We investigated the relation between the direction of droplet movement and the centroid of thermal cells. The angle between the vector from the center of the droplet to the centroid of all cells and the velocity vector are almost 180º with respect to time [Fig. 1(c)], and the more cells distributed on the opposite side of the direction of droplet motion and their size is bigger, likely those pushing the droplet to move forward.

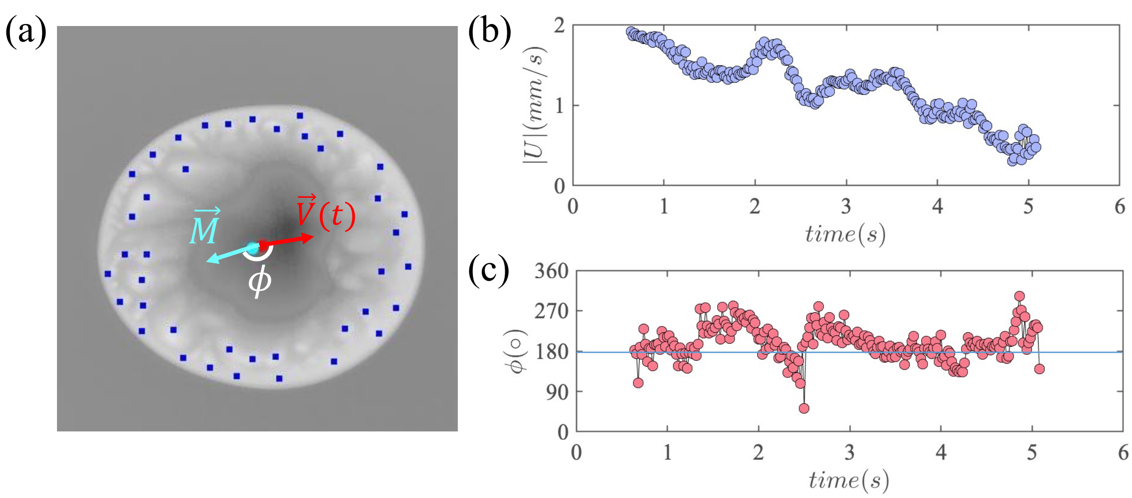


Figure 1. (a) Thermal pattern on the droplet interface. The droplet was deposited on the sapphire surface heated to 55°C. (b) The corresponding instantaneous velocity of the moving droplet with time. (c) Evolution of angle between two vectors. **V**(t) is the velocity vector of the moving droplet and **M**(t) is the vector between the centroid of all thermal cells and the droplet center position.

REFERENCES

[1] Deegan, R. D., Bakajin, O., Dupont, T. F., Huber, G., Nagel, S. R., & Witten, T. A. (1997). Capillary flow as the cause of ring stains from dried liquid drops. *Nature*, 389(6653), 827-829.

[2] Ristenpart, W. D., Kim, P. G., Domingues, C., Wan, J., & Stone, H. A. (2007). Influence of substrate conductivity on circulation reversal in evaporating drops. *Physical review letters*, 99(23), 234502.

Impact of Boiling Liquids: Droplets on surfaces

Bernardo Palacios Muñiz, Devaraj Van Der Meer

University of Twente

ABSTRACT

The impact of droplets onto surfaces is a broad and well explored subject in the case that droplet and surrounding atmosphere are thermally decoupled. However, the physics of impacting droplets that are in thermal equilibrium with their own vapor remains unexplored, although these type of impacts are precisely what is encountered during transport of cryogenic liquids, where sloshing wave impact is responsible for most of the load experienced by the containing structure. For this work we have developed an experimental setup to create boiling droplets and observe their impact upon solid surfaces. The setup consists of a sealed chamber at reduced pressure, where a liquid (Novec™ 7000) is allowed to evaporate until saturation. The rest of the liquid fills a reservoir and is then transported to a needle, from which saturated droplets are generated and allowed to fall onto a temperature controlled sapphire substrate. So, we are in fact creating boiling droplets surrounded by their own vapor. Heat exchanging channels allow to control the temperature of the chamber and therefore the saturation pressure of the liquid. The temperatures of the substrate, chamber and liquid reservoir and the pressure of the chamber are measured accurately and continuously to provide the experimental conditions and allow for control and repeatability. We present first results from experiments at different impact velocities and substrate temperatures.

REFERENCES

Indicates here the references

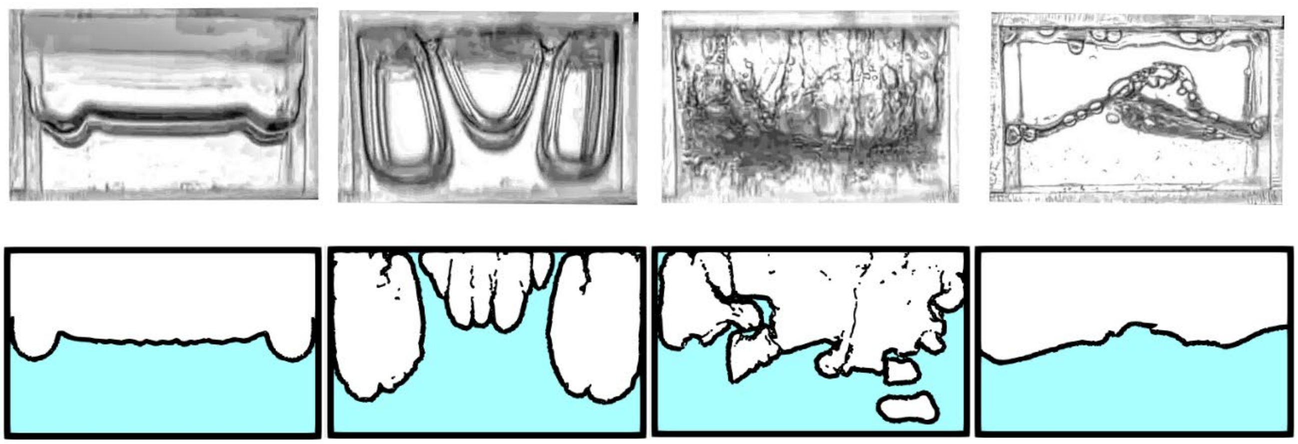
Experimental and numerical characterization of violent sloshing flows. A single degree of freedom approach to the problem.

Jon Martinez-Carrascal
L M. González-Gutiérrez
Javier Calderon-Sanchez
Universidad Politécnica de Madrid, Madrid, Spain

ABSTRACT

In this work, some of the most fundamental aspects of an aeronautical sloshing problem will be presented using a simplified model. This model consists of a single degree of freedom version of the original problem which keeps the essence of the fluid structure interaction and the most relevant physical aspects of the industrial case. Two different approaches have been followed: on the one hand, an experimental rig has been designed to measure and visualize different magnitudes of the problem; on the other hand, two smoothed particle hydrodynamics (SPH) formulations, single phase and multiphase, have been adapted and coupled to a damped mass-spring system to obtain a local representation of the flow interaction with the tank movement.

A complete non-dimensional analysis of the problem in terms of additional damping has been performed, and the dependency on the most relevant non-dimensional relations has been monitored. A coupled numerical simulation where both the tank and the fluid are combined has been used to study the system, and their results are compared to the experiments. The SPH models, extensively validated in the sloshing literature, are used to calculate the magnitude and frequency of the vertical force between the fluid and the tank. The extra dissipation of the tank's mechanical energy caused by the fluid action is quantified for a particular configuration with constant filling level and a wide range of non-dimensional numbers. The sensitivity of the extra damping to the variation of the non-dimensional numbers is evaluated, and the most relevant ones are compared to the equivalent experimental tests. Results show that the numerical tool developed is able to capture the different phenomena involved and can be used to determine the influence of the different phenomena happening in violent vertically excited flows.



REFERENCES

Martinez-Carrascal, J.; Gonzalez-Gutierrez, L.M. Experimental study of the liquid damping effects on a SDOF vertical sloshing tank. Journal of Fluids and Structures 2021, 100, 103172.

Marrone, S.; Colagrossi, A.; Gambioli, F.; González-Gutiérrez, L. Numerical study on the dissipation mechanisms in sloshing flows induced by violent and high-frequency accelerations. I. Theoretical formulation and numerical investigation. Phys. Rev. Fluids 2021, 6, 114801. doi:10.1103/PhysRevFluids.6.114801. II. Comparison against experimental data. Phys. Rev. Fluids 2021, 6, 114802. doi:10.1103/PhysRevFluids.6.114802.

Martinez-Carrascal, J.; Calderon-Sanchez, J.; González-Gutiérrez, L.M.; de Andrea González, A. Extended computation of the viscous Rayleigh-Taylor instability in a horizontally confined flow. Phys. Rev. E 2021, 103, 053114. doi:10.1103/PhysRevE.103.053114.

Calderon-Sanchez, J.; Maritnez-Carrascal J., Gonzalez-Gutierrez, L.M.; A., C. A global analysis of a coupled violent vertical sloshing problem using an SPH methodology. Engineering Applications of Computational Fluid Mechanics 2021.

Vertical sloshing phenomena in rectangular tanks

D. Ubaldini[1], D. Rossi[1], S. Di Giorgio[2], S. Pirozzoli[1] and A. Iafrati[2]
[1] Dipartimento di Ingegneria Meccanica e Aerospaziale,
Sapienza Università di Roma, via Eudossiana 18, Rome, 00184, Italy
[2] CNR-INM, Istituto di Ingegneria del Mare, Via di Vallerano 139, Rome, 00128, Italy

The dynamic behaviour of liquids in moving containers is a fascinating subject that has attracted the attention of geophysicists and seismologists, engineers, mathematicians and other scientific workers for a period of many years. The excitations or tank motions that may lead to liquid sloshing are also quite varied, encompassing a wide system of directions, amplitudes and frequencies, while the more classical Lateral sloshing or Rotational sloshing are intensively studied, the case of Vertical sloshing has still been little investigated.

Vertical slosh dynamics is one of the possible dynamics of the fluid stowed in the tanks of an aircraft and, when it occurs, it exhibits different characteristics compared to the classic sloshing. This type of sloshing is well known to provide a noticeable increase in the overall structural damping, in particular the dissipative effect on a moving rigid tank under harmonic excitation has been studied with numerical simulations and the results are compared with experimental studies (Saltari et al., 2022) (see fig. 1). An incompressible two-phase Navier-Stokes solver with a Volume-of-Fluid (VOF) method for the tracking of the interface is used (Pirozzoli et al., 2019), and a series of two- and three-dimensional simulations, varying amplitude and frequency of tank accelerations have been carried out. The two-dimensional simulations show a very good agreement with experimental results in computing dissipated energy for all those cases where the 3D effects were limited, while it failed where a more chaotic regime was present. A couple of 3D simulations have been performed in order to improve the computing of the dissipation energy in the cases with the three-dimensional effect can not be neglected. Sloshing vertical dynamics were more deeply studied through a Fourier analysis on the motion of the centre of gravity of the fluid and by defining some parameters in order to characterize fluid-air mixing.

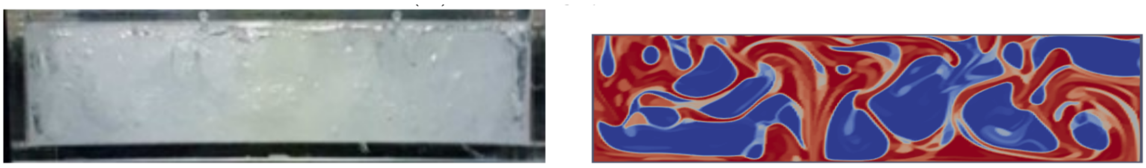


Figure 1: Snapshots of experimental sloshing behaviour (left column) and simulations (right column) at different values of acceleration amplitude and frequency

References

S. Pirozzoli, S. Di Giorgio, and A. Iafrati. On algebraic TVD-VOF methods for tracking material interfaces. *Comput. Fluids*, 189:73–81, 2019. ISSN 0045-7930.

F. Saltari, M. Pizzoli, G. Coppotelli, F. Gambioli, J. E. Cooper, and F. Mastroddi. Experimental characterisation of sloshing tank dissipative behaviour in vertical harmonic excitation. *Journal of Fluids and Structures*, 109:103478, 2022. ISSN 0889-9746.

Accurate CFD estimation of dynamic loads in liquid containment structures:

Towards a high-fidelity simulation platform for cryogenic systems

Leon Malan, Bevan Jones, Yusufali Oomar, Arnaud Malan and Stephane Zaleski
Industrial CFD Research Group, Dept. Mechanical Engineering , University of Cape Town

ABSTRACT

Fluid dynamic response inside tanks undergoing acceleration is of interest in many practical applications, including aircraft fuel tank design [1] and liquified natural gas (LNG) [2] transport systems. We present an overview of progress in the development of finite volume CFD based numerical methods to calculate tank dynamic loads for industrial applications. More specifically, a weakly compressible gas formulation[3] is applied with a one-fluid model using an algebraic Volume-of-Fluid method to calculate slosh loads. Simulations are performed on vertically excited tanks and compared to experiment [4]. An example snapshot of a 2D simulation is shown in Figure 1 below. Simulation results compare the evolution of slosh loads as well as system dissipation to experimental measurements. In the numerical method, we perform an in-depth mechanical energy analysis on the fluid to study the dissipation mechanisms of the system from the sloshing fluid.

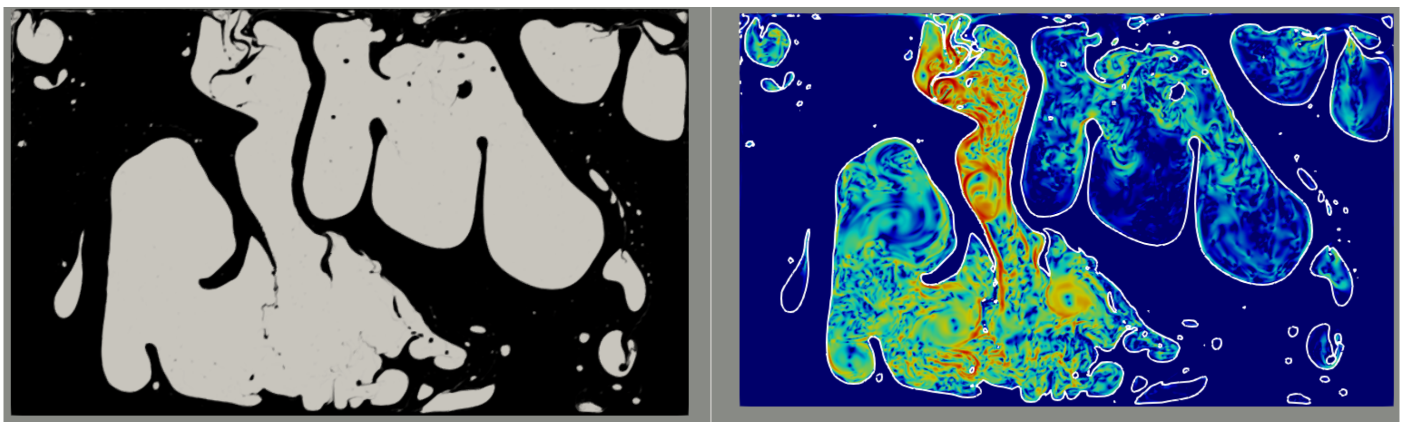


Figure 1: Simulation snapshot of a vertically excited tank containing oil and air. On the left the two fluids are shown after 0.3s with the oil in black and the air in light grey. On the right the interface is shown with a white line and the colour sale represents the turbulent viscosity ratio, as calculated using a Smagorinsky-Lilly LES turbulence model. The range is from 0 (blue) to 2 (red).

We then proceed to comment on recent developments to extend this platform to include physical models required in a cryogenic tank environment. These include conjugate heat transfer between tank wall and fluid, natural convection and mass transfer (phase change). Some preliminary results are presented with prospects on further work.

REFERENCES

[1] F. Gambioli and A. Malan, Fuel loads in large civil airplanes, in International Forum on Aeroelasticity and Structural Dynamics IFASD 2017, Como, Italy 25-28 June 2017 (ELPO - Editore, 2017), Vol. 4, pp.2641–2660.
[2] Ancellin, M., Brosset, L. and Ghidaglia, J.M., 2018, June. Numerical study of phase change influence on wave impact loads in LNG tanks on floating structures. In *International Conference on Offshore Mechanics and Arctic Engineering* (Vol. 51302, p. V009T13A026). American Society of Mechanical Engineers.
[3] Heyns, J.A.; Malan, A.G.; Harms, T.M.; Oxtoby, O.F. A weakly compressible free-surface flow solver for liquid–gas systems using the volume-of-fluid approach. J. Comput. Phys. 2013, 240, 145–157, doi:10.1016/j.jcp.2013.01.022.
[4] Martinez-Carrascal, J. and González-Gutiérrez, L.M., 2021. Experimental study of the liquid damping effects on a SDOF vertical sloshing tank. *Journal of Fluids and Structures*, *100*, p.103172.

Influence of ullage pressure on wave impacts induced by solitary waves in a flume tank: findings from SLING project

[1]*R. Ezeta,* [2]*L. Kimmoun, and* [3]*L. Brosset*
[1]Maritime Research Institute Netherlands (MARIN), Haagsteeg 2 6708PM, Wageningen, The Netherlands
[2]École Centrale Marseille, 49 rue Frédéric Joliot-Curie13384, Marseille, France
[3]Gaztransport & Technigaz (GTT), 1 route de Versailles 78470, Saint-Rémy-lès-Chevreuse, France

ABSTRACT

SLING is a research Programme that started in 2016 aiming at better understanding the physical phenomena related to wave impacts in LNG (Liquified Natural Gas) membrane tanks on floating structures and the biases induced by the use of sloshing model tests in order to assess sloshing loads. The consortium is composed of four Dutch Universities, research institutes, classification societies and industrial partners. The project was funded by NWO, the Dutch Research Council, and industrial partners.

In this framework, a new facility, named *The Atmosphere* (or ATM), was designed and built in MARIN. It consists of a 10 m-long wave flume inside an autoclave. A piston-type wavemaker located at the end of the flume enables the accurate generation of different types of water waves. At the other end of the flume, a vertical wall is instrumented with 100 pressure transducers in order to capture the wave impact loads. The autoclave provides a precise control of the water and gas temperature and enables the possibility to vary the composition of the ullage gas by any mixture of air, Nitrogen, Helium, Sulfur Hexafluoride and water vapor.

A recent wave impact test campaign was performed in the ATM. Solitary waves were studied in combination with a bathymetry profile - a beach made of two inclined planes located in front of the impact wall - in order to favor the wave breaking in front of the end wall.

By adjusting the wavemaker prescribed signal in order to vary the amplitude of the solitary wave, we generated smooth and repeatable breaking waves, with Nitrogen as ullage gas at ambient conditions. The piston motion leading to a large-gas-pocket impact was repeated for increasing values of the ullage pressure p_u, from 0.1 bar to 8 bar. These environmental conditions were obtained by injecting progressively more Nitrogen into the autoclave, keeping the temperature close to 20°C. By doing so, both the gas-to-liquid density ratio and the gas compressibility varied at the same time.

We find that the variations of the global wave shape just before the impact are insignificant for $p_u \in [0.1, 1.0]\ bar$. For increasing p_u (larger than 1 bar), we observed a progressive backward inclination of the wave as already observed in sloshing model tests [1] and in numerical simulations [2]. This phenomenon is explained by the increase of gas-to-liquid density ratio which leads to an increase of the energy transfer from the liquid to the gas. Moreover, for increasing ullage pressures, we observed an earlier development of free surface instabilities leading to larger local perturbations of the wave shape – especially at the crest level. For the largest p_u (8 bar), this leads to a spray around the crest.

For $p_u \in [0.1, 1.0]\ bar$, as the variations of the global wave shape are insignificant, the comparisons of the gas pocket behavior and of the maximum pressure at the crest level for the different p_u is relevant.:

- As described in many studies throughout the literature, we observed that the gas pocket oscillations are quicker and the maximum pressure within the pocket is larger for increasing p_u, as the gas stiffness becomes larger. The gas pocket behavior is well described by the well-known Bagnold model.
- From the few repetitions of the tests made for each p_u, we observed a decrease of the average maximum pressure at crest for increasing p_u. This trend is attributed to larger local perturbations of the wave shape due to free surface instabilities.

For $p_u \in [1.0, 8.0]\ bar$, since the wave shapes differ significantly, the maximum pressure at crest cannot be fairly compared. Similarly, the initial sizes of the entrapped gas pockets differ and therefore the comparison of their behaviors with regard to p_u is biased. Nonetheless, we provide recommendations on how this could be tackled in future works.

The pressure maps obtained for the different studied wave impacts show clearly the different areas corresponding to the Elementary Loading Processes (ELP) described in [3]. The pressure signals are smooth and seem very relevant as reference for numerical simulations.

REFERENCES

[1] M. R. Karimi, L. Brosset, J.-M. Ghidaglia and M.-L. Kaminski (2016), *Effect of ullage gas on sloshing – Part II: local effects of gas-liquid density ratio*. European Journal of Mechanics B/Fluids, Vol. 57, pages 82 - 100, doi: 10.1016/j.euromechflu.2015.11.011.

[2] S. Etienne, Y.-M. Scolan and L. Brosset (2018). *Numerical Study of Density Ratio Influence on Global Wave Shapes Before Impact*, 37th Int. Conf. on Ocean, Offshore and Artic Eng. (OMAE), Madrid (Spain), Juin 2018.

[3] Lafeber, W., H. Bogaert, and L. Brosset (2012). *Elementary Loading Processes (ELP) involved in breaking wave impacts: findings from the Sloshel project*. 22nd Int. Offshore and Polar Eng. Conf. (ISOPE). Rhodes, Greece. Vol. 3, pp 265-276.

Wave Impact dynamics of breaking waves at phase transition conditions: findings from SLING project

[1]*R. Ezeta,* [2]*L. Kimmoun, and* [3]*L. Brosset*
[1]Maritime Research Institute Netherlands (MARIN), Haagsteeg 2 6708PM, Wageningen, The Netherlands
[2]École Centrale Marseille, 49 rue Frédéric Joliot-Curie 13384, Marseille, France
[3]Gaztransport & Technigaz (GTT), 1 route de Versailles 78470, Saint-Rémy-lès-Chevreuse, France

ABSTRACT

SLING is a research Programme that started in 2016 aiming at better understanding the physical phenomena related to wave impacts in LNG (Liquified Natural Gas) membrane tanks on floating structures and the biases induced by the use of sloshing model tests in order to assess sloshing loads. The consortium is composed of four Dutch Universities, research institutes, classification societies and industrial partners. The project was funded by NWO, the Dutch Research Council, and industrial partners.

Inside a LNG membrane tank on a floating structure, both phases of the fluid are in a state close to the thermodynamic equilibrium – leading to intricate multiphase phenomena, especially during wave impacts, which is not fully accounted for in the literature [1-2].

In this talk, we show experimental results of wave impacts under phase transition conditions for various gas-to-liquid density ratios, i.e. $DR = \rho_g/\rho_l$, with ρ_g and ρ_l the gas (vapor) and liquid densities respectively. The experiments were carried out in the Atmosphere (ATM) facility in MARIN [3] with water and water vapor as the liquid and gaseous phase respectively, where we were able to perform experiments at values of the temperature T and ullage pressure p_u along the vapor pressure of water, namely within $DR \in [1.7 \times 10^{-5}, 4.0 \times 10^{-3}]$ ($T \in [20, 167]\,°C$). The breaking waves used in this study were generated with solitary waves in combination with a bathymetry profile in order to induce breaking. We captured the wave shape upon impact with two high-speed cameras and measured the impact loads with an array of 100 pressure transducers that allowed us to extract pressure maps. Specifically, we looked at the vapor pocket that is entrapped when a breaking wave impacts on a solid wall (see figure below). Our experiments have revealed that – when the DR is small (low temperatures) – a Rayleigh-type collapse of the vapor pocket takes place, which is accompanied by a short-duration large amplitude pressure peak. As the DR increases, we observed that this effect is mitigated and the impacts resemble more the case of the oscillation of non-condensable gas pockets. We elucidated on the possible mechanisms behind these phenomena by placing our experiments in the context of a theoretical model of a vapor pocket that takes into account both pocket dynamics (Rayleigh-Plesset equation) and heat transfer (Plesset-Zwick formula).

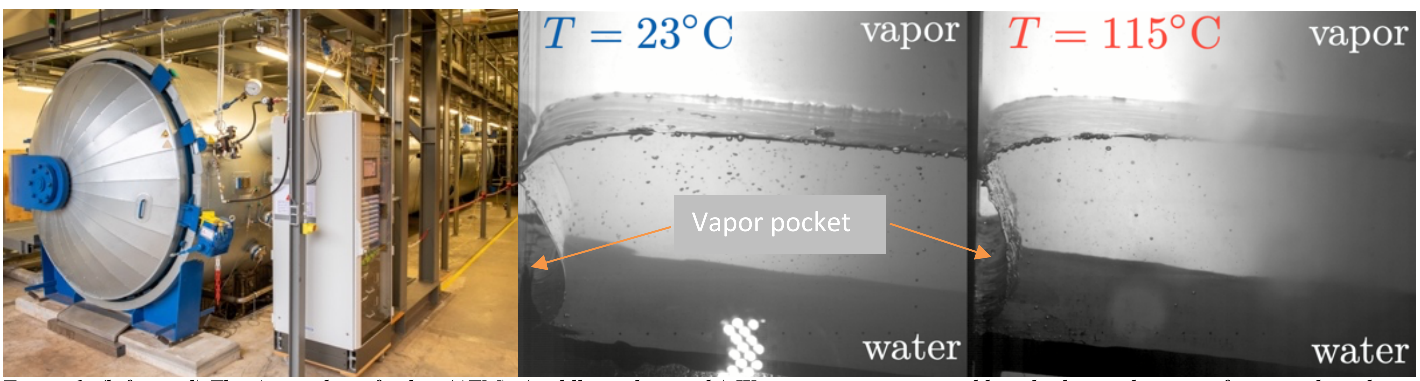


Figure 1. (left panel) The Atmosphere facility (ATM). (middle, right panels) Wave impact as captured by a high-speed camera for two selected temperatures.

REFERENCES

[1] Bogaert H., An experimental Investigation of Sloshing Impact Physics in Membrane LNG Tanks on Floating Structures (2018). PhD Thesis. TU Delft.
[2] S. Maillard, L. Brosset. Influence of density ratio between liquid and gas on sloshing model test results (2016). ISOPE 19, 271-279.
[3] Novakovic et al., Study of Global Wave Repeatability in the New Multiphase Wave Lab (MWL) (2020). ISOPE I-20, 3244.

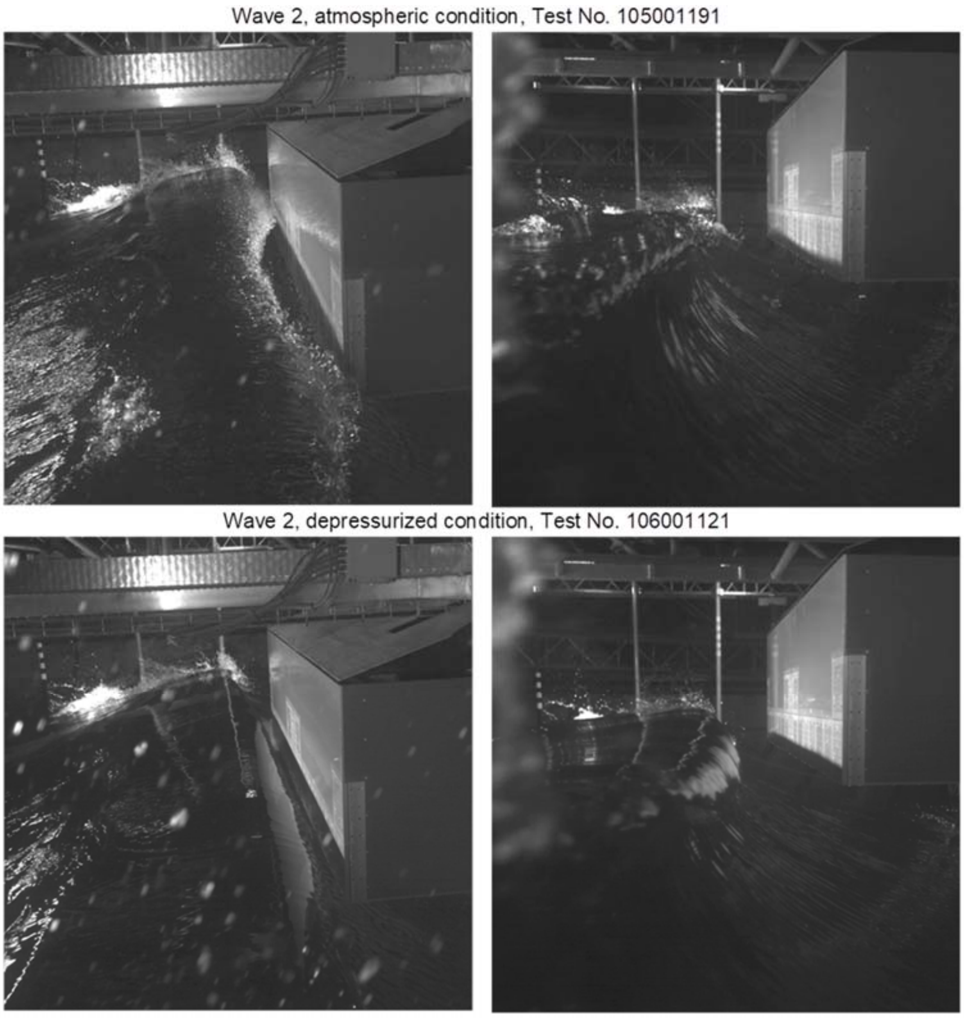
Elementary loading processes and scale effects involved in wave-in-deck type of impact loading

Jule Scharnke & Joop Helder
MARIN

ABSTRACT

To quantify the loading due to breaking waves, model tests are currently the option of choice. However, it is suspected that scale effects lead to an overestimation of the prototype loading. As such, a typical question in impact load assessment is how conservative load measurements are. To understand how realistic model testing is in this respect, it is necessary to quantify scale effects in the measured loading due to breaking waves as well as to investigate to which extent entrapped air in the wave and during the wave impact is involved. To do so, the BreaKin Joint Industry Project was started in 2016. The objective of the BreaKin JIP was to get more insight into scale effects involved in wave-in-deck model tests and to take first steps towards linking wave kinematics with measured impact loads.

During this JIP wave-in-deck model tests were carried out in MARIN's Depressurized Wave Basin (DWB) at two scales (25 and 50) in atmospheric and depressurized condition. The presented work gives an overview of the results of the BreaKin JIP regarding loading processes involved in wave-in-deck type of impacts, effects of depressurization on measured loads and possible sources of scale effects.



Simulation of a Sloshing Wave Using a Pressure-Based Multiphase Compressible-Incompressible Flow Solver

João Muralha[a], Luís Eça[a], Christiaan M. Klaij[b]
[a]IST Técnico Lisboa, Lisbon/Portugal; [b]MARIN Maritime Research Institute Netherlands, Wageningen/The Netherlands

ABSTRACT

A large part of the numerical simulations performed in naval and offshore applications assume incompressible fluids. But in the case of sloshing, slamming or cavitation events, compressibility effects may play an important role. To include these effects in ReFRESCO (www.refresco.org): first, a single-phase compressible solver was implemented, verified, and tested (Muralha et al., 2020; Muralha et al., 2021). And only then the multiphase solver was developed (Muralha et al., 2022). The solver uses a finite volume approach with cell-centred collocated variables, and multiphase flows are modelled using the Volume of Fluid technique. The liquid phase is considered incompressible, and a polytropic law is for the compressible gas phase.

To gain insight into the effect of considering the gas phase of a multiphase flow compressible, a tank motion that results in an entrapped gas-pocket is simulated. The test case is a tank used for an experimental benchmark organized by ISOPE (Loysel et al., 2013), with a filling ratio of 20%. The movement is a combination of surge and pitch motions. The pressure at the tank left wall (Figure 1) and the free surface position at different time instants (Figure 2) were selected as quantities of interest to illustrate the results.

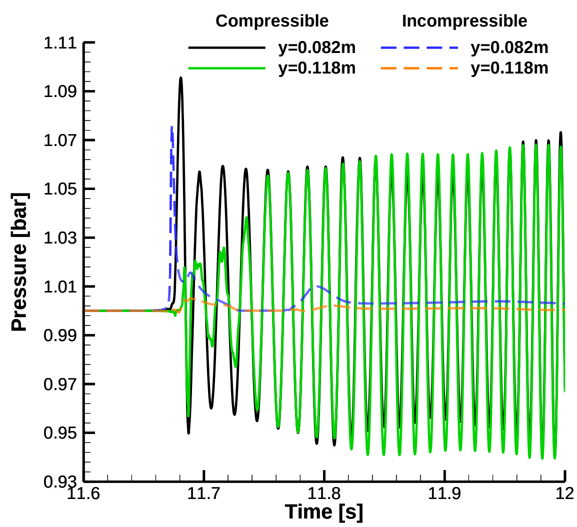


Figure 1 - Pressure time history at two different heights at the left domain wall for simulations with (solid lines) and without (dashed lines) compressibility effects lines

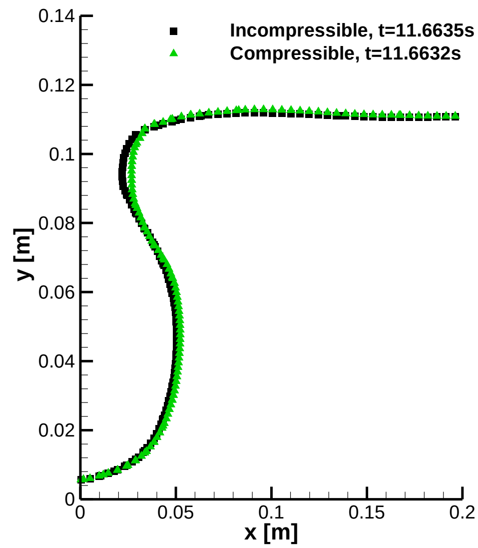


Figure 2 - Free surface position for the simulations with (green triangles) and without (black squares) compressibility effects

REFERENCES

Loysel, T., Gervaise, E., Moreau, S. and Brosset, L., (2013). "Results of the 2012-2013 Sloshing Model Test Benchmark". *Proceedings of 23rd International Offshore and Polar Engineering Conference (ISOPE)*, Anchorage, (AK), USA, Vol. 3, pp 141-152.

Muralha, J., Eça, L., and Klaij, C. M. (2020). "Code Verification of a Pressure-Based Solver for Subsonic Compressible Flows". *Journal of Verification, Validation and Uncertainty Quantification*, Vol. 5(4)

Muralha, J., Silva, C., Eça, L., and Klaij, C.M., (2021) "Solution Verification Studies of a Pressure-Based Compressible Flow Solver". *Proceedings of 14th WCCM-ECCOMAS Congress*, Virtual Congress.

Muralha, J., Eça, L. and Klaij, C. M., (2022). "On the Formulation of a Pressure-Based Multiphase Compressible-Incompressible Flow Solver". Submitted to *Proceedings of 24th Numerical Towing Tank Symposium (NuTTS)*, Croatia.

Hydroelastic slamming in pure and aerated water

Simon Tödter, Jens Neugebauer, Ould el Moctar
University of Duisburg-Essen, Institute of Ship Technology, Ocean Engineering and Transport Systems

ABSTRACT

Vessel sailing in heavy seas are prone to impact loads, which may lead to structural vibrations and damages. These loads vary in space and time and are highly dynamic. In the present study systematic experimental investigations of impact loads on rigid and flexible structures entering into pure and aerated water were performed. During experiments test bodies were guided and were driven by a device to ensure constant velocity water impact. Forces, pressures and strains were measured to analyze the influence of hydroelasticity and aeration. For aerated tests tomographic sizing was used to reconstruct air bubbles and to quantify the void fraction and the bubble size distribution. Previous studies showed that for rigid impacts already a small void fraction yields significant damping of impact loads. Here, we investigate the combined effect of hydroelasticity and aeration.

3D Hydroelastic impact on axisymmetric bodies

Malenica Š.[(1)], Seng S.[(1)], Diebold L.[(1)], Scolan Y.M.[(2)]
[(1)]BUREAU VERITAS, Marine & Offshore Division, Research Department
[(2)] ENSTA Bretagne

ABSTRACT

Evaluating the structural response under the strong hydrodynamic impact loading is still extremely challenging task and no efficient nor robust numerical method exists for general applications (slamming, sloshing, ditching, underdeck impact…). The engineering practice consists in defining the equivalent pressure loading and apply it on the selected part of the structure in a quasi-static manner. At the end of the simulations, the response is sometimes corrected using some empirical dynamic amplification factor. The exact definition of the equivalent design pressure and the associated dynamic amplification factor raise some issues and strongly depends on the particular application. In some cases the design pressures are deduced from the experiments in spite of large scatter in the measurements and in other cases the CFD is used in spite of the lack of convergence when evaluating the extreme impact pressures.

The consistent way to deal with this problem would be to use fully coupled hydroelastic modelling procedures. When using those methods, the local extreme impact pressures will be, when necessary, naturally “filtered” by the structure, however their effects (sometimes minor) will be taken into account through the hydroelastic coupling procedure. The purpose of the present work is to validate one of such methods which is based on coupling the OpenFOAM CFD model on loading side with the FEM model on structural response side. Different interaction regimes (impulsive, dynamic and quasi static) are considered and the method is validated by comparisons with the semi-analytical model of Scolan (2004). Once validated, the model is applied to practical simulations of sloshing impact in the tanks of LNG carriers.

REFERENCES

[1] Faltinsen O.M., 2000. “*Hydroelastic slamming.*” J Mar Sci Technol Vol. 5, pp. 49–65
[2] Khabakhpasheva T. & Korobkin A.A., 2013. : “*Elastic wedge impact onto a liquid surface: Wagner’s solution and approximate models.*”, J. of Fluids and Structures, Vol. 36 pp. 32–49
[3] Korobkin A.A., 1985.: “*Initial asymptotics of solution of three-dimensional problem on a blunt body penetration into ideal liquid*”, Dokl. Akad. Nauk SSSR 283, 838- 842.
[4] Leissa, A. W., 1973.: “*Vibration of shells*” Scientific and Technical Information Office, National Aeronautics and Space Administration, Vol. 288.
[5] Malenica S., Gatin I., Seng S., Jagite G., Diebold L., Khabakhpasheva T. & Korobkin A.A., 2021.: “*Some aspects of the local hydro-structure interactions during hydrodynamic impacts*”, 23rd Numerical Towing Tank Symposium (NUTTS), Duisburg, Germany.
[6] Seng S., 2013.: “*Slamming and whipping analysis of ships*”, PhD thesis, Technical University of Denmark.
[7] Seng S., Jensen J.J. & Malenica S. 2014.: “*Global hydroelastic model for springing and whipping based on a free-surface CFD code (OpenFOAM)*”, International Journal of Naval Architecture and Ocean Engineering, Vol. 6, No. 4
[8] Soedel, W., 2004.: “Vibrations of Shells and Plates.” CRC Press.
[9] Wagner H., 1932.: “*Uber Stoss - und Gleitvorgange an der Oberflache von Flussigkeiten.*”, ZAMM 12, pp 193–215.
[10] Zhao R. & Faltinsen O., 1992.: “*Water entry of two-dimensional bodies.*” J. Fluid Mech., 246, pp 593–612.
[11] Malenica S., Seng S., Diebold L., Scolan Y.M., Korobkin A. A. & Khabakhpasheva T., 2022.: “On three dimensional hydroelastic *impact*”, 9th International Conference on Hydroelasticity in Marine Technology (HYEL), Roma, Italy.

Monitoring, Mapping, Tracking Wave Driven Rocks and Boulders

Constantinos Melenaou, Lucia Robles Diaz, Daniel Toal and Frederic Dias
University College Dublin, University of Limerick and ENS Paris-Saclay

ABSTRACT

We investigate the wave-driven motion of small rocks and boulders, with emphasis on the seabed.

The aim is to obtain a better understanding of the effects of waves on coastal boulder deposits and coastal erosion. Firstly, numerical methods are being used to simulate and predict wave-structure interactions. The method currently being used is Smoothed Particle Hydrodynamics, a fully Lagrangian method ideal for the fluid structure interactions. Two different solvers have been selected for this project, namely SPhinXsys and DualSPHysics. Differences between the two solvers are investigated in order to evaluate their performance to this specific problem. Simulations are carried out in combination with field experiments to further improve our understanding of boulder motions in a more realistic manner. For the experimental field work, surveys will be conducted to determine the boulders field on the seabed at target locations with high wave activity due to wave breaking. A preliminary set of experiments was conducted during the week of 13-17 June 2022.

Numerical study of surface tension and viscosities influence on global wave shapes before impact

Zeineb Horriche, Stéphane Etienne, Cédric Béguin
Ecole Polytechnique de Montréal, Montréal, Canada
Laurent Brosset
GTT (Gaztransport & Technigaz), Saint-Rémy-lès-Chevreuse, France

ABSTRACT

Loads induced by sloshing in Liquefied Natural Gas (LNG) membrane tanks on floating structures are always assessed thanks to sloshing model tests usually performed at scale 1:40. The model tank is filled with water and a mixture of gases selected in order to match the same gas-to-liquid density ratio as in the LNG tank. It is instrumented with numerous pressure transducers and installed on the platform of a hexapod. The motions of the floating structure are computed at scale 1 for all conditions that it is expected to experience throughout its life. These motions are down-scaled to scale 1:40 assuming a Froude similarity and then imposed to the hexapod.

The properties of the fluids inside the model tank cannot match those that would be necessary to get a complete similarity with the flow inside the LNG tank. For instance, the surface tension at small scale is about 800 times too large with regard to the right value in order to match the Weber similarity. The kinematic viscosity of both liquid and gas are about 250 times too large with regard to the right values in order to match the respective Reynolds similarities.

The objective of the current work is to study numerically the influence of such large variations of Weber and Reynolds numbers on the global wave shape of a Single Impact Wave (SIW) just before impact, even just before the compressibility of the gas matters. The studied SIW is generated in 2D with water and air in a 20 m long wave canal. The water is initially at rest with a free surface presenting a large bump close to one of the vertical walls. A quarter of ellipse bathymetry located in front of the other vertical wall favors the wave breaking. This wave leads to a large entrapped gas pocket after impact. It has already been studied by different authors ([1], [2] for instance)

We could have chosen to study this SIW at different length scales, for instance 1:1, 1:10, 1:20 and 1:40, applying Froude similarity (it corresponds here to scaling the geometry of the tank and the initial wave shape) and keeping the same fluids, water and air, at the different scales. This approach would have led to the targeted variations of Weber numbers and Reynolds numbers but all together. Therefore, we preferred to maintain the same scale while modifying either the surface tension or the viscosities.

The reference case is at scale 1:40 with water and air. Two parametric studies have been conducted:

- Three calculations at scale 1:40 with surface tension varying in accordance with Weber numbers that would be obtained at scales 1:1, 1:10 and 1:20 with water and air. The other characteristics of the fluids are those of water and air;
- Three calculations at scale 1:40 with viscosities of both liquid and gas varying in accordance with Reynolds numbers that would be obtained at scales 1:1, 1:10 and 1:20 with water and air. The other characteristics of the fluids are those of water and air.

Using CADYF, a bi-fluid high-fidelity front-tracking software developed by Ecole Polytechnique Montreal to simulate separated two-phase incompressible viscous flows with surface tension, the global wave shapes and the variation of the total energy composition in the system are compared.

Apart from its stabilization effects which lead to less developed free surface instabilities for Weber numbers corresponding to lower scales, the work necessary to deform the free surface turned out to explain the main part of the loss of mechanical energy when cumulated in both the liquid and the gas. The remaining part corresponds almost to the calculated viscosity dissipation within the liquid and the gas. The numerical dissipation takes finally only less than 10^{-4} of the initial total energy. We have now a full picture of the transfer of energies between liquid, gas and free surface and we are expecting to compare it with results from direct length scale variations.

A numerical comparison study between CADYF and ComFLOW; a Cartesian-grid based Volume-of-Fluid CFD simulation code, is ongoing and has shown several points of similarity for a reference case and we are hoping to expand it to cover the above-mentioned studies.

REFERENCES

[1] Guilcher, P.-M. , Couty, N., Brosset, L. and Le Touzé, D., (2013). *Simulations of breaking wave impacts on a rigid wall at two different scales with a two-phase fluid compressible SPH model*, Int. Journal of Offshore and Polar Eng. Vol. 23, n° 4, 241-253.

[2] S. Etienne, Y.-M. Scolan and L. Brosset (2018). *Numerical Study of Density Ratio Influence on Global Wave Shapes Before Impact*, in Proc. of the 37th (2018) Int. Conf. on Ocean, Offshore and Artic Eng. (OMAE), Madrid (Spain), Juin 2018.

Numerical modelling of variability in liquid impacts

R. A. Remmerswaal, A. E. P. Veldman
University of Groningen

ABSTRACT

Understanding the physics involved in breaking wave impacts is essential for optimising the shape of cargo containment systems used to store liquefied natural gas (LNG) inside LNG carriers. In particular, the role of free surface instabilities, that occur at the wave tip before impact and result in significant variability of impact pressures, is not well understood. The numerical modelling of breaking wave impacts can bridge the knowledge gap that results from the experimental study of liquid impact problems, which is done at small scale and therefore overestimates the stabilising role of surface tension.

Several improvements on state-of-the-art volume of fluid methods for two-phase flow are proposed.

- A sharp and structure preserving method is proposed for the transport of mass and momentum, resulting in exact mass and momentum conservation. It is moreover ensured that the convective operator does not contribute to the change in kinetic energy in the semi-discrete limit.
- A piecewise parabolic reconstruction of the interface is proposed [1], which alleviates the observed lack of convergence of the interface curvature in time-dependent problems when traditional piecewise linear interface reconstruction methods are used. This is essential for the modelling of capillarity, as a lack of curvature convergence implies the presence of spurious kinetic energy via the imposition of the Young-Laplace equation.
- Finally, a model for the interface shear layer is proposed which allows for a velocity discontinuity in the interface tangential direction(s). This implies that the resolution requirements due to viscosity, which are considerable for the high Reynolds number flow problems that we consider here, can be significantly reduced.

The proposed methods are validated using numerous academic test cases, and are subsequently applied to the simulation of liquid impact problems. We demonstrate the ability to numerically model (both in two as well as three spatial dimensions) free surface instabilities at the wave tip before impact, where we show that the resulting instability wavelength is in agreement with linearised potential flow theory. In particular, the development of a Kelvin-Helmholtz instability is observed, and we demonstrate in 3D the development of a spanwise Rayleigh-Taylor instability on the wave tip, which has previously been observed in experimental results.

This work is part of the research programme SLING, which is (partly) financed by the Netherlands Organisation for Scientific Research (NWO).

REFERENCES

[1] Remmerswaal, R. A. and Veldman, A. E. P. (2021). Parabolic interface reconstruction for 2D volume of fluid methods. *arXiv preprint arXiv:2111.09627.* (Accepted in *Journal of Computational Physics*.)

Coherent vortical structures generated by breaking waves

S. Di Giorgio[1], S. Pirozzoli[2] and A. Iafrati[1]
[1] CNR-INM, Istituto di Ingegneria del Mare, Via di Vallerano 139, Rome, 00128, Italy
[2] Dipartimento di Ingegneria Meccanica e Aerospaziale,
Sapienza Università di Roma, via Eudossiana 18, Rome, 00184, Italy

The flow generated by the breaking of free-surface waves in a periodic domain is simulated numerically by means of a gas-liquid multiphase Navier-Stokes solver. The solver relies on the Volume-of-Fluid (VOF) approach, and interface tracking is carried out by using a novel algebraic scheme based on a tailored TVD limiter (Pirozzoli et al., 2019). The solver is proved to be characterized by low numerical dissipation, thanks to the use of the MAC scheme, which guarantees discrete preservation of total kinetic energy in the case of a single phase. Both two- and three-dimensional simulations have been carried out, and the analysis is presented in terms of energy dissipation, air entrainment, bubble fragmentation, statistics and distribution. Particular attention is paid to the analysis of the mechanisms of viscous dissipation. For this purpose, coherent vortical structures (Horiuti and Takagi, 2005), are identified and the different behaviour of vortex sheets and vortex tubes are highlighted, at different Re. The correlation between vortical structures and energy dissipation demonstrates clearly their close link both in the mixing zone and in the pure water domain, where the coherent structures propagate as a consequence of the downward transport. Notably, it is found that the dissipation is primarily connected with vortex sheets, whereas vortex tubes are mainly related to flow intermittency.

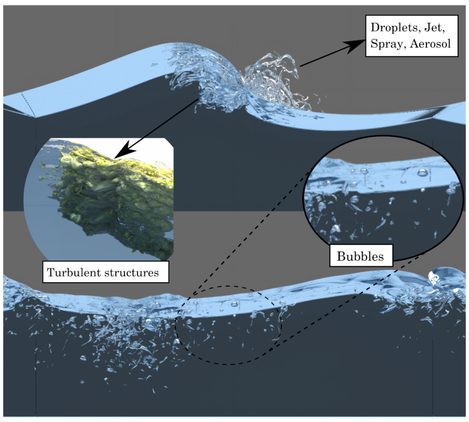


Figure 1: Rendering of the breaking wave process, in which bubbles, droplets, sprays and turbulent structures are highlighted.

References

K. Horiuti and Y. Takagi. Identification method for vortex sheet structures in turbulent flows. *Phys. Fluids*, 17(12):1–4, 2005. ISSN 10706631. doi:10.1063/1.2147610.

S. Pirozzoli, S. Di Giorgio, and A. Iafrati. On algebraic tvd-vof methods for tracking material interfaces. *Comput. Fluids*, 189:73–81, 2019. ISSN 0045-7930. doi:10.1016/j.compfluid.2019.05.013.

Modeling and simulation of the multiphase pre-neutralizer reactor for NPK fertilizers production

Mohamed HAMOU [1], Lhachmi KHAMAR [1,2], Saad BENJELLOUN [1], Jean-Michel GHIDAGLIA [1,3]

[1] MSDA Modeling, Simulation, and Data Analysis, Mohammed VI Polytechnic University, Benguerir, Morocco
[2] LIPIM, Laboratoire d'Ingénierie des Procédés, Informatique et Mathématiques, ENSA Khouribga, Sultan Moulay Slimane University, Khouribga, Morocco
[3] CMLA Centre des Mathématiques et leurs Applications, ENS Cachan, Paris Saclay University, Paris, France

ABSTRACT

The use of NPK fertilizers greatly impacts the yields of agricultural production and helps to meet the increasing food needs of the growing world population. These granular NPK fertilizers are produced in a wet granulation process using successively stirred tank pre-neutralizers and rotating drum granulators.

In the multiphase pre-neutralizer reactor, the liquid phosphoric acid reacts with the gaseous ammonia to produce a slurry mono/diammonium phosphate salt (MAP/DAP) that feeds the drum granulator. The slurry properties such as density, viscosity, composition, temperature, etc. depend on the operating conditions inside the pre-neutralizer and on the raw materials (phosphoric acid and ammonia) quality.

The modeling and simulation of the pre-neutralizer is a powerful tool that helps to predict the slurry properties and therefore to optimize the granulation step where the slurry granulates into solid MAP/DAP fertilizers pellets.

In this work, we have modeled the industrial pre-neutralizer reactor using a phenomenological modeling approach. We have accounted for all the physical, chemical, thermodynamic, rheological, etc phenomena occurring in the equipment. Each phenomena is modeled separately as a sub-model (see figure 1), and the set of the sub-models are connected between them (using mass and heat balance equations) to build up the global reactor model. This approach based on the hierarchical sub-model decomposition is very efficient to handle the multi scales and the complexity of the system. The model equations are programmed and solved using the open source software OpenModelica (see figure 2). Both steady state and dynamic simulations were carried out to assess the reactor behavior during steady state regimes and during startups and shutdowns. The solver that has been used to solve the model algebraic differential equations is the DASSL solver [1]. Multiphase 3D computational fluid models [2] and experimental measurements/ industrial data were used to estimate and to identify the parameters for this phenomenological model.

The simulation results we have obtained compared very well with the industrial plant DCS data gathered at different operating conditions (see table 1). The pre-neutralizer outlet product (MAP/DAP slurry) properties can accurately be predicted using our phenomenological white-box model, and what/if scenarios simulations can be carried out to assess the system behavior and to find the optimal operating conditions/ parameters that should be adopted depending on the raw materials quality. To the best of our knowledge and based on the literature review, our work is a first of its kind. A phenomenological and first principles high-fidelity model is developed for the study and the optimization of the granulation process (the pre-neutralizer and the drum granulator).

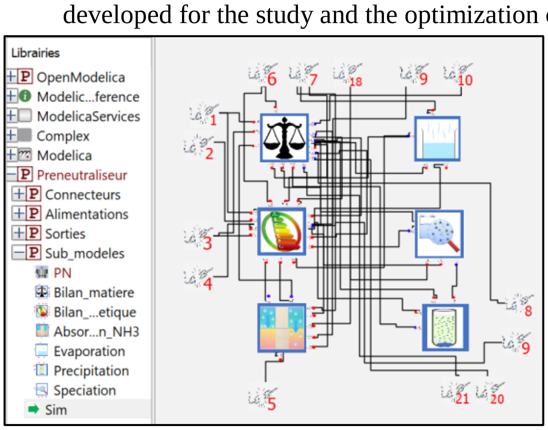


Figure 1. The pre-neutralizer sub-models connected between them

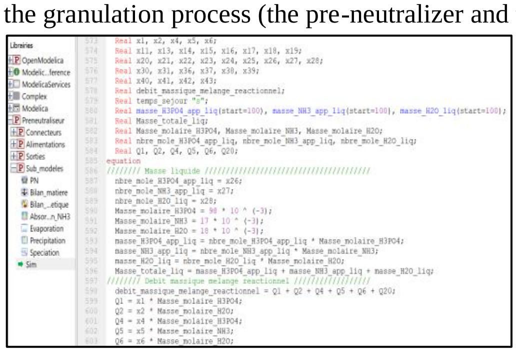


Figure 2. The pre-neutralizer model equations programmed in OpenModelica environment

MAP/DAP slurry properties	Simulation results	Plant data	Gap (%)
temperature (°C)	131	127	3
Flowrate (kg/h)	64211	66000	3
H_3PO_4/ NH_3 ratio	1,54	1,52	1

Table 1. Example of comparison between simulation results and plant data

REFERENCES

[1] https://www.openmodelica.org/doc/OpenModelicaUsersGuide/latest/solving.html
[2] Elmisaoui, S., et al. "Modeling and Study of Hydrodynamic flow within the Preneutralizer Reactor using CFD Approach. " Computer Aided Chemical Engineering. Vol. 48. Elsevier, 2020. 103-108.

Multiscale modeling of granulation process using DEM-PBM coupling

Achraf NAGIHI[1], Mohamed HAMOU[1], Khalil FERRADI[1], Safae EL MISAOUI[1], Saad BENJELLOUN[1], Lhachmi KHAMAR[1]

[1] University of Mohamed VI Polytechnique, Lot 660, Hay Moulay Rachid, Ben Guerir 43150,

Morocco

Abstract

The control and optimization of the granulation of phosphate fertilizers (DAP, MAP, TSP, etc.) requires a good understanding of the phenomena and processes taking place within the granulator. In this work we develop mathematical models capable of describing these phenomena.

A multi-scale and hybrid model is developed using a two-way coupling approach between Discrete element methods (DEM) and Population Balance Methods (PBM) using Data-driven surrogate models. DEM and PBM are two modeling techniques widely used in the field of fertilizer granulation. This hybrid model considers the collision frequencies of particles and the distribution of liquids providing a suitable framework for the complex sub-processes of wet granulation.

The multi-scale approach in this work consists on using the results of the fine 3D and intensive DEM simulations to develop a behavioral (surrogate) model capable of accurately predicting collision frequencies at a macro scale [1], as well as other phenomena such as breakage [3, 4] and viscosity and liquid film effects. This approach has the advantage of avoiding the use of very heavy calculations each time in DEM calculations (LIGGGHT) to capture heterogeneities within a granulation process. Indeed, DEM simulations are executed for certain operating parameters obtained from a complete factorial plan (Design of Experiments). The intermediate points within the DOE are interpolated using models based on the data. Using this approach, a step towards Quality by Design (QbD) can be taken for a better modeling and control of the granulation process.

Key words

Rotating drum granulator, granulation, PBM, DEM, coalescence, breakage, kernels.

References

[1] Dana Barrasso Ashutosh Tamrakar Rohit Ramachandran (2015), Model Order Reduction of a Multi-scale PBM-DEM Description of a Wet Granulation Process via ANN.

[2] Ahmad B. Albadarina Timothy & D Lewisb Gavin M. Walkera (2017). Granulated polyhalite fertilizer caking propensity.

[3] Manel Gargouri, Chaker Chtara, Patrick Sharrock, Ange Nzihou, Hafed El Feki, (2012). Experimental Study of the Purification of an Industrial Fertilizer (Mono-Ammonium Phosphate) toLarger Scale Using an Experimental Design.

[4] L.E. Patruno, C.A. Dorao , H.F. Svendsen , H.A. Jakobsen (2008). Analysis of breakage kernels for population balance modelling.

Dynamic modeling of large phosphates slurry pipeline systems in `Modelica`

Saad Benjelloun[1], Ahmed Ja[1], Jean Michel Ghidaglia[1,2].

[1] MSDA, Université Mohammed VI Polytechnique, Morocco.
[2] Centre Boreli, Ecole normale supérieure Paris-Saclay, France.

June 23, 2022

Abstract

Since 2014, OCP group, a global leader in the phosphate industry, has decided to transport phosphates ore via a slurry pipeline network spanning more than 187 kilometers with highly variable topography. In this work, we model the industrial-scale, 187km-long, phosphate slurry piping system using the system modeling framework and the `Modelica` language.

A homogenized one dimensional Navier-Stokes model with varying density, is used with a finite volume discretization scheme, while Newtonian and non-Newtonian rheological behaviour of the slurry is accounted for.

Given the network's size, fully 3D flow models are too complex to build and consume too much CPU time to be useful in real-time process control. However, fine and intensive 3D CFD computations were used to build a surrogate rheological model for the slurry [1]. The resulting model has been tested against real data and is able to predict the operating parameters for continuous flow insurance for different slurry types and process water.

References

[1] Marwane Elkarii, Radouan Boukharfane, Saad Benjelloun, and Chakib Bouallou. Surrogate model for slurry pipe flow pressure drop prediction using cfd. In *Proceedings of the 32nd European Symposium on Computer Aided Process Engineering (ESCAPE 32), June 12-15th, 2022, Toulouse, France*, 2022.

Assessment of RANS and LES-SGS Turbulence Models on the Flow Hydrodynamics Prediction in an Industrial Stirred Chemical Reactor

Safae Elmisaoui[1,2], Radouan Boukharfane[1], Lhachmi Khamar[1,3], Jean-Michel Ghidaglia[1,4]

[1]*Mohammed VI Polytechnic University, MSDA Program, Lot 660, Hay Moulay Rachid, Benguerir 43150, Maroc*
[2]*Mohamed V University, Laboratoire du génie civil et environnement, Avenue le Prince Héritier, Salé 11060, Maroc*
[3]*Sultan Moulay Slimane University, LIPIM, ENSA Khouribga, BP 77 Boulevard Beni Amir, Khouribga 25000, Maroc*
[4]*Centre Borrelli, ENS Paris-Saclay, CNRS, Université Paris-Saclay, 94235 Cachan, France*

ABSTRACT

Different turbulence models have been developed and used to predict the hydrodynamics and turbulence in stirred vessels, each one of them demonstrated its effectiveness and robustness in some applications. This work aims at evaluating the performance of both Reynolds averaged Navier-Stokes (RANS) and Large-Eddy Simulation (LES) turbulence closures of a stirred reactor [1]. The age-old CFD modeling method based on the Reynolds Averaged Navier-Stokes (RANS) approach is still considered the most employed turbulence modeling approach due to its acceptable accuracy and affordable computational cost for predicting the hydrodynamics involving complex geometries, but with the development in the HPC and computer technology, the use of the hybrid Detached Eddy Simulation (DES) model, which blends the RANS approach with LES founds its applications [2,3]. The DES methodology shows its increasing potential and accuracy in flows prediction more than the RANS models. Especially for its informative ability to predict the flow regimes in the moving zone characterizing the impeller rotation. This work carried out a three-dimensional simulation of a stirred sparged chemical reactor for a preneutralization reaction. The commercial software ANSYS FLUENT 2020 R2 is employed to model the gas-liquid flow based on an Eulerian multiphase approach. The model validation is conducted by comparing results of the simulated scenarios with experimental data from literature. Furthermore, numerical simulations based on both RANS and LES paradigms of mixing in a full-scale preneutralizer chemical reactor are performed for a further improvement in the process effectiveness with the aim of assessing the influence of a various number of turbulence closures.

REFERENCES

[1] Hartmann, H., Derksen, J. J., Montavon, C., Pearson, J., Hamill, I. S., and Van den Akker, H. E. A. "Assessment of Large Eddy and RANS stirred tank simulations by means of LDA " Chemical Engineering Science, 59, no. 12 (2004): 2419-2432.

[2] Alonzo-Garcia, A., Mendoza-Escamilla, V. X., Martinez-Delgadillo, S. A., Gonzalez-Neria, I., Gutiérrez-Torres, C. D. C., & Jiménez-Bernal, J. A. (2019). On the performance of different RANS based models to describe the turbulent flow in an agitated vessel using non-structured grids and PIV validation. Brazilian Journal of Chemical Engineering, 36, 361-382.

[3] Yeoh, S. L., G. Papadakis, and Yianneskis, M. "Numerical simulation of turbulent flow characteristics in a stirred vessel using the LES and RANS approaches with the sliding/deforming mesh methodology." Chemical Engineering Research and Design, 82, no. 7 (2004): 834-848.