

SIMULATION OF SHIP FLOODING IN OPEN FOAM: A PRAISE FOR OPEN SOURCE CFD

Pedro P. S. P. Lopes¹, Henrique M. Gaspar², Marcos Maturana¹, Marcelo R. Martins¹, and Kazuo Nishimoto¹

¹Depto of Ocean and Naval Eng., University of São Paulo, Brazil.
pedro.paludetto@labrisco.usp.br

²Depto. of Ocean Operations and Civil Eng., NTNU, Norway. henrique.gaspar@ntnu.no

KEYWORDS

Open Source, OpenFOAM, CFD, Flooding Simulation.

ABSTRACT

This paper presents an overview of a finite-volume CFD Opensource tool and its applications in industry and academia. A study case of a cruise ship flooding simulation is presented to show its capacity. To ensure the reliability of the results, a grid convergence analysis was performed and compared with benchmark data provided by FLARE. This European consortium provided open data on flooding benchmarks. Additionally, general arrangement layout modifications were tested. The main corridor width was changed, and water elevation probes were placed in different rooms to evaluate the change in the corridor width. Results showed that when the corridor width increases, the time to flood for the rooms far from the breach is reduced. This behavior was opposed to the room near the breach, requiring a longer time to fill the room as the corridor width increased. The study case showed that studies with small variations could be important for ship design purposes, for instance, and doing that in experimental facilities is expensive, showing the necessity of good numerical free and open-source tools, such as OpenFOAM.

A CFD OVERVIEW TOWARDS AN OPEN AND SCALABLE FREE-SOURCE TOOL

The modeling of fluids and heat is challenging and has been a concern for society since we were organized as such. Archimedes was maybe the first great thinker on hydrodynamics, from 287 BC to 212 BC, when he studied hydrostatics to determine how to measure densities and volumes (Netz, 2022). In the 15th Century, Leonardo Da Vinci contributed with his flow study presented in a nine-part treatise called *Del moto e misura dell'acqua* (de Vinci, 1828), where he presented sketches of the flow motion in 54 Tavolas, which covers water surfaces, movement of water, water Communications of the ECMS, Volume 38, Issue 1, Proceedings, ©ECMS Daniel Grzonka, Natalia Rylko, Grazyna Suchacka, Vladimir Mityushev (Editors) 2024 ISBN: 978-3-937436-84-5/978-3-937436-83-8(CD) ISSN 2522-2414

waves, and eddies.

In the late 17th Century, Sir Isaac Newton postulated the three Newton Laws, and his main contributions to fluid dynamics lie in the 2nd law, Newtonian's viscosity concept, and the reciprocity principle. The 18th and 19th centuries presented several important fluid dynamic works, such as the Bernoulli equations from Daniel Bernoulli, Euler equations from Leonhard Euler, and many others, until Claude Louis Marie Henry Navier and George Gabriel Stokes proposed, not together, viscous effect on Euler equations, and is known nowadays as Navier-Stokes (NS) Equations, the basis for the modern-day computational fluids dynamics (CFD), Lemarié-Rieusset (2018) presented a nice overview of NS-equations and its story. Those equations are so complex to solve for real flow problems that only with the advent of modern digital computers in the sixties could they be solved for simplified problems.

The advent of modern digital computers leads to the first commercial CFD codes. In 1972, Imperial College colleagues Brian Spalding and Brian Launder started the first commercial code with a k-epsilon turbulence model for industrial uses. They founded Concentration Heat and Momentum Ltd (CHAM) and developed several CFD codes apart. In 1981, they combined all the codes into the Parabolic Hyperbolic Or Elliptic Numerical Integration Code Series (PHOENICS), considered the first commercial general-purpose CFD package. In a relatively short period after, in 1982, the first version of Fluent was launched with a more robust code, able to calculate combustion, dispersed phases, and natural convection (Awel, 2008). Several new software were created, some merged, and some disappeared. Nowadays, great companies such as Ansys and Siemens own some of the most used codes in the industry, such as Fluent and StarCCM+.

Since the start of the internet, free source software has been available. In the CFD world, free source software is developed mainly by the academy in universities. Some of them can be mentioned such as *deal.ii*, an open-source finite element library maintained by Texas A& M University, Clemson University, and Universität Heidelberg; SU2, an opensource C++-based

library to perform PDE analysis from Stanford University; Méfisto, from the Université Pierre and Marie Curie in France, which is a 3D finite element method based. There are infinite free source libraries to solve fluid dynamics problems. With the growth of collaborative and online version-control repositories (GitHub) and object-oriented programming, plenty of software has become available for academia.

At the beginning of the 1990s, at the Imperial College, one of the most traditional centers of CFD research, the development of OpenFOAM started. This package of libraries for finite volume tools was released to the public in 2004 with the GNU general public license, which was a breakthrough in the CFD world and the most widely used CFD tool nowadays. (Chen et al., 2014)

OPENFOAM: A BRIEFLY OVERVIEW

OpenFOAM (OF) constitutes a C++ CFD toolbox for customizing numerical solvers. There are over 60 different solvers there, performing simulations of all kinds, e.g., basic CFD, combustion, turbulence modeling, electromagnetic, heat transfer, multiphase flow, stress analysis, and many others. Also, for being an open-source code, several research groups assemble their own specialized tools based on the libraries. Another important feature of OF is the capacity to do parallel computing, making complex simulations capable of being solved in supercomputers, for instance.

This CFD tool is basically a set of numerical tools based on the Finite Volume Method (FVM) to solve Partial Differential Equations (PDE). OpenFOAM does not have a Graphic User Interface (GUI) and is based on a set of directories and C++-based text files. For flow visualization, data analysis, and mesh generation, the code is dependent on third-party software, such as Paraview for flow visualization, Python for data analysis, and snappyHexMesh or cfMesh for grid Generation. All of the tools mentioned are also free-source. It can also be linked with commercial tools, with special attention paid to commercial mesh generators. OpenFOAM has tools to translate mesh formats from Fluent Mesh and StarCCM mesh generator to OF-friendly format, with the tools *fluent3DMeshToFoam* or *star4ToFoam*, for instance. Several tools like this are available over OpenFOAM repositories.

A generic model of directory organization on openFoam is shown in Figure 1.

Those folders and text files are mandatory for any simulation using OF. On the folder *System*, general settings are chosen. The control parameters are presented in the *controlDict* text file, such as information about time discretization, outputs, and Courant Number. In the files *fvSchemes*, the user sets discretization schemes for gradients, Laplacian, and rotational operators, and the *fvSolution* sets the linear algebra for the discretized and linear systems.

On the folder *Constant*, the file *Properties* is responsible for the definition of properties parameters, such as viscosity, gravity, turbulence coefficients, etc.

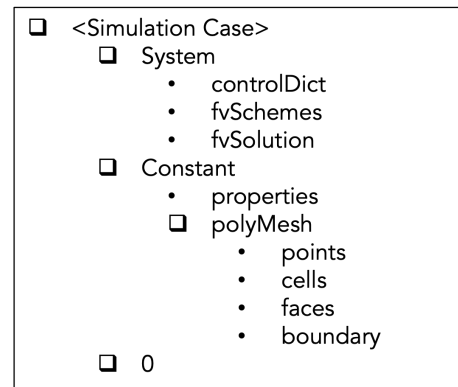


Figure 1: Folders Organization.

The folder *polyMesh* is responsible for keeping all the mesh information. Finally, in the folder *0*, all the information regarding initial and boundary conditions is stored. Depending on the simulation's complexity, different folders or configuration files can be added for each solver.

DATASETS FOR HYDRODYNAMICS IN FLOODING SCENARIOS

Passenger vessel flooding has been researched since the Titanic accident in 1912. The British government's first international regulation stated the first requirements for safe navigation, watertight, fire-resistance, and life-saving appliances (Wheeler, 1914). In 1987, a RoPax vessel capsized in Belgium, resulting in 193 casualties. After this accident, it was necessary to understand the dynamic behavior of a ship while capturing water on deck. After SOLAS 1990, IMO adopted new standards for new vessels regarding stability parameters. In 1994, the most deadly capsized accident occurred in Estonia when a RoPax ship capsized, killing 850 passengers. The leading cause of the accident was excessive water accumulation on the main deck. After 1999, successive benchmarking studies on quantitative risk assessment were carried out for renowned research institutes.

Benchmarks

The first benchmark, HARDER, from 1999 to 2003, analyzed accident data for collision. SAFEDOR (Spanos and Papanikolaou, 2009), GOALDS (Papanikolaou et al., 2012), and EMSA III (Spouge and Skjong, 2014) updated the analysis of accident data for collision and grounding, improving the risk model for flooding. The eSAFE (Atzamos et al., 2019) study combined collision, bottom, and side grounding hazards based on EMSA III high-risk models. Finally, FLARE (Ruponen et al., 2022) revised the high-level risk models, leading to a new risk structure. This benchmark project developed a new open accident database used to validate the numerical model proposed here.

FLARE: Open Data

Within the FLARE benchmark, Ruponen et al. (2022) presented the main results of the FLARE study. The main goal of this study was to evaluate different numerical models from other institutes in a blind test.

The results were compared with an experiment performed by MARIN on a cruise ship model with six floodable decks and 82 internal openings. They tested three case tests: large breach in calm water, large breach in waves, and small breach in calm water. Most of the numerical solvers used a hydraulic approach using Bernoulli's equation, in which two solvers used the inclined plane for the floodwater surface, and five solvers used the horizontal plane, i.e., did not capture the variation of water high inside the rooms. Only one solver used CFD with the volume of fluid method (VoF) to model the flooding.

In conclusion, most of the cases captured the dynamic behavior of the vessel well enough. This study did not evaluate the dynamic behavior of the flooding inside the rooms, requiring further investigation. Also, on the FLARE project, a preliminary study tested a captive one-deck model in calm water, (Ruponen et al., 2021). All the data is available on the FLARE website, including 3D CAD models, and was used for the validation of the case study proposed by this paper.

OPEN CFD FOR FLOODING

Flooding as Case for Mapping Open Software and Open Data

Besides the benchmarking studies, several authors investigated the flooding phenomena. Parsons et al. (2008) proposed an Intelligent Ship Arrangements (ISA), which provides an optimization technology and design tool to aid ship design considering intelligent decision-making support. Its model considers the US Navy standards and rational measures of merit that would permit comparisons between arrangement designs. Additionally, Daniels and Parsons (2008) presented a formulation of a hybrid-agent algorithm, which considers a genetic algorithm for allocating spaces to Zone-decks optimization. Lee et al. (2023) proposed a study that merged general arrangement design and accident simulations. He developed a conceptual analysis of alternative designs to minimize the consequences of maritime autonomous surface ship MAAS accidents. Their paper aimed to present design alternatives and emergency response systems to appropriately control emergencies and reduce the impact of accidents such as flooding and fire on board. Moreover, simulations using the flooding simulation tool SURVSHIP Lee (2015) were carried out for the ship model, considering two cases, one with heating venting and air conditioning (HVAC) systems on the bottom and the other with HVAC on the ceiling. This study evaluated a general arrangement with flooding accidents.

Among those efforts to understand flooding phenomena, few studies used open-source CFD as a tool. Only

DNV simulations used OF as a code on the FLARE benchmark and were not used for the proposed set of cases.

STUDY CASE

This paper proposes a study case assessment of the influence of the general arrangement in a cruise vessel on flooding situations. A baseline model was developed based on the FLARE benchmark study from Ruponen et al. (2021) to reach this goal. A grid convergence analysis was performed, and the results were compared with the benchmark data.

Simplification and Assumptions

Every CFD simulation goal is to model the most high-fidelity case possible. A perfect model would deal with the turbulence as a Direct Numerical Simulation (DNS), which solves every flow scale. For a three-dimensional, multiphase, and transient flow, DNS simulation would require an extremely fine grid, and even with the more developed supercomputer available, this would probably still not be possible. Nevertheless, plenty of turbulence models can be considered in a simulation like that, which would not require so much computational power. However, as mentioned in Ruponen et al. (2022), laminar flow is considered a reasonable assumption once the simulations are performed in model scale and will be considered in this assessment.

Another important simplification is the fixed platform consideration. The dynamic behavior of the flooding ship plays an important role in the vessel's stability during the accident. Nonetheless, this work is a preliminary study of the GAs influence on the flooding process and will be focused on understanding how the flow fills up the rooms for different general arrangements (GAs). It will be recommended as future work on the GA influence on the flooding dynamics.

The results were evaluated in specific points proposed by FLARE in Ruponen et al. (2021). Figure 2 shows the five probes where the water elevation was measured. The layout is a typical deck arrangement of cruise ships and was proposed by FLARE.

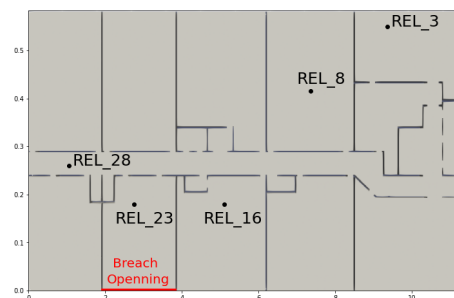


Figure 2: Probes Location and Breach Opening.

OF does not have a built-in function to evaluate water elevation. Hence, a field analysis must be done. To extract the water elevation on the probe locations, a combination of Python and Paraview was used to find

the height of the parameter $\alpha = 0.5^1$ over time. This postprocessing takes a reasonable amount of computer processing time, up to 8 hours, to evaluate in a machine with 64 GB of RAM memory.

Domain, Mesh and Boundary Conditions

Four decks were simulated to test the GA influence, varying the main corridor width. The baseline case corridor is about 7.5 % of the deck width. The assessment tested the effect of the corridor width increase by 10%, 12.5%, and 15%, as presented in Figure 3.

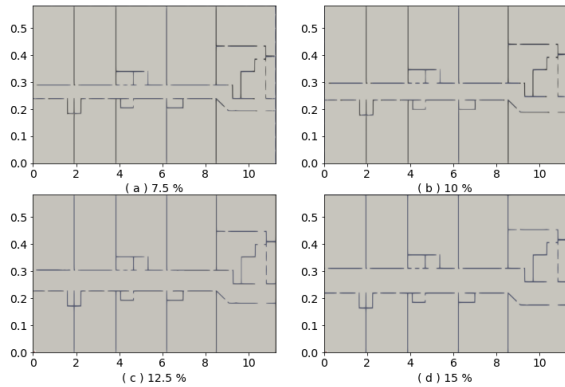


Figure 3: General Arrangement Setups

The meshes were generated using the libraries *blockMesh* and *snappyHexMesh* and the open data available in the FLARE database. Figure 4 shows the baseline case mesh. All the other GA meshes were generated with the same parameters. The boundaries were kept, and the corridor width was increased. Hence, the room area evolved smaller as the corridor width increased.

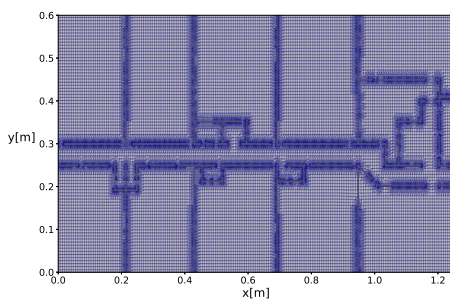


Figure 4: Baseline Case Mesh

A proper boundary condition (BC) in a CFD model is of huge importance. On the one hand, a wrong BC will lead to wrong results; this is the first source of errors in a simulation called GIGO (*garbage in, garbage out*). On the other hand, a high-fidelity BC in complex cases can make the simulation computationally costly. The present study case is a good example of it. In a

¹This parameter is part of the VoF method to model the free-surface in *interFoam* solver. Its explanation is beyond this paper's scope; more about the method can be found in Katopodes (2019).

high-fidelity BC, a flooding simulation would place the geometry in a virtual tank and make it flood. This would add millions of cells to the tank, the turbulence would impose an important role in the simulation, and the laminar condition assumption would not be proper. Additionally, making the ship flood would imply the use of complex methods for moving the grid.

As shown in Figure 2, the control volume of the model was only the ship layout. The flow inlet was modeled as a Dirichlet BC of constant velocity at the breach opening. The inlet velocity of the flow was calculated by the averaged hydrostatic pressure corrected by the discharge coefficient recommended by the benchmark data.

Another challenging BC modeling was related to the top surface. In high-fidelity modeling, the modeling of the roof as a wall would be ideal. As the ship floods, the air will be compressed on the top, and the compressibility would have to be modeled. This compressibility is responsible for numerical instability, and a finer mesh would be necessary. So, to make the simulation feasible, a Neumann BC was modeled as a null pressure gradient on the top surface. The walls were modeled as a non-slip condition, and the laminar flow assumption helped save time consumption using coarser grids.

Model Validation

In order to properly evaluate the numerical model, a Validation and Verification (V&V) methodology is required. There are plenty of methodologies available. In a general way, verification is a numerical uncertainty estimation and is totally linked to the numerical model. Three main uncertainty sources must be evaluated: a) iterative errors related to the numerical iterative process. This error can be mitigated by setting thresholds as low as possible so that the iteration can move forward; b) round-up errors, which can be avoided by using double precision for the calculations; c) discretization errors are the hardest to predict and the most important in terms of uncertainty accumulation. These errors are the results of the discretization methods in numerical cases. Its source can be from space, time, or a combination of both. In simpler words, those errors are related to the poor discretization of the domain, and generally, the user needs to choose the larger discretization as possible to save computational time. Eça and Hoekstra (2009) proposed a method to evaluate the monotonic behavior of a three-dimensional surface fit using Richardson extrapolation. The surface is formed by a cloud of measured parameters in function of time and space discretization, $f(s, dt)$, where s stands for grid size and dt for time step size. If the surface has a monotonic pattern, the infinity limit is the "real" value, and the uncertainty is calculated as a percentual of each parameter for the grid and time discretization chosen. A detailed explanation with examples can be found in Lopes (2019).

The validation of the results is to compare the nu-

merical results with their uncertainties with the experimental or real data. Some authors support that the uncertainties related to the experiments should be considered in this analysis. (Wang et al., 2021)

Nevertheless, numerical uncertainties are expensive to achieve in terms of computational power. As the present study case aims to show the effectiveness of Open-source software for hydrodynamics problems, a simplified grid convergence analysis was performed. Three grids were tested for the baseline case and compared with the simulations and experiments available data from FLARE. The results of this step are presented in Figure 5, related to REL 23 in Figure 3, Figure 6, related to REL28 probe, Figure 7, related to REL16 probe, Figure 8, related to REL8 probe and Figure 9, related to REL3 probe.

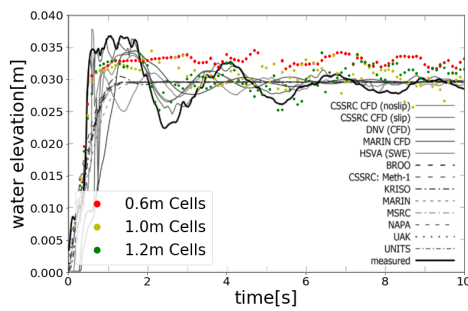


Figure 5: Grid Convergence for Probe Location REL 23

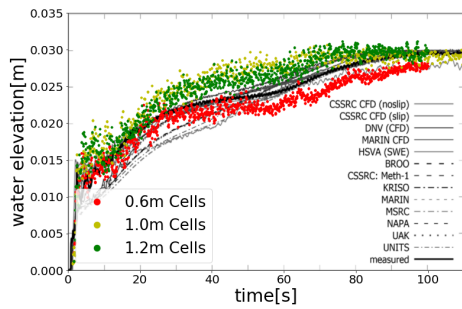


Figure 6: Grid Convergence for Probe Location REL 28

In all the cases, the grid with 0.6 million of (M) elements showed poor agreement with the FLARE data. If we look at the time that it takes for the room to be filled, the coarser grid takes considerably more time to fill. On the other hand, the 1 M cell grid presented a faster filling pattern if compared with the benchmark data and the finer grid with 1.2 M cells. The finer grid showed consistent agreement with the benchmarking data, being then considered for the general arrangement assessment further on.

Results and Discussions

To conclude this case study, the results of the general arrangement assessment are presented in Figures 10,

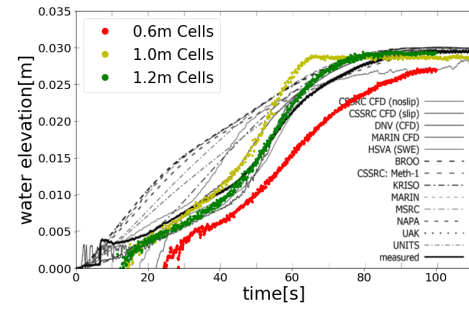


Figure 7: Grid Convergence for Probe Location REL 16

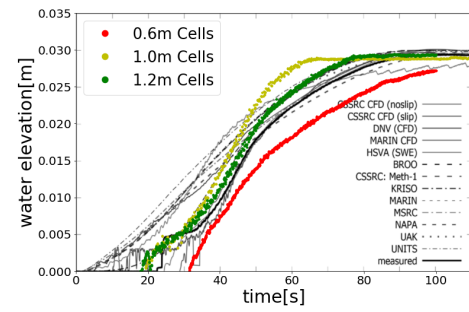


Figure 8: Grid Convergence for Probe Location REL 8

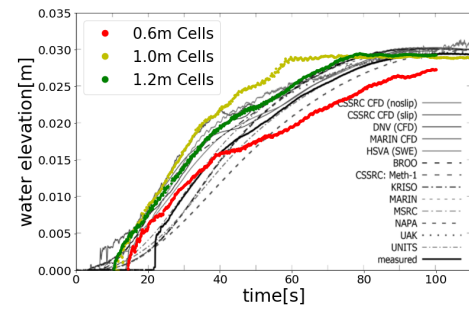


Figure 9: Grid Convergence for Probe Location REL 3

11, 12, 13, and 14. The results are presented over the benchmark results in a black and white background in order to have a reference for the results.

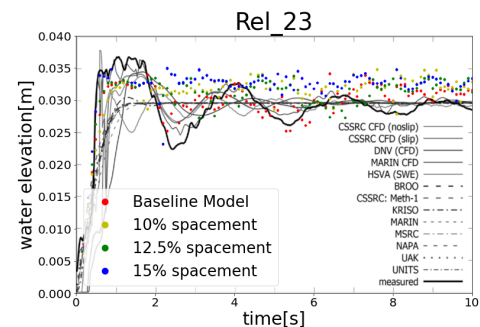


Figure 10: Grid Convergence REL 23

In Figure 10, the probe 23 is evaluated. This probe is located near the breach opening. Therefore, it is the first room to fill with water completely. Just 10 seconds were analyzed, and once after that, no considerable variations occurred. At this position, the room is filled in less than one second. High fluctuations are detected due to the high velocity of the flow due to the proximity to the breach. No conclusions can be taken regarding the corridor width.

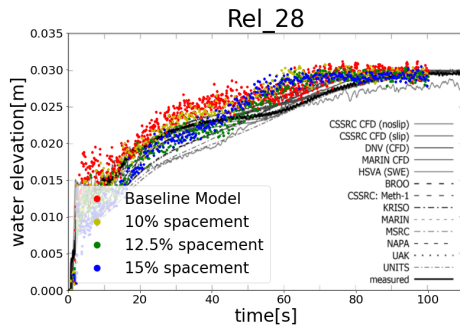


Figure 11: General Arrangement Variation - REL 28

At the probe, REL28 (Figure 11), located just after the first room in the corridor, a noise curve rises due to the high velocities in this area. A slowed filling is observed if compared with the REL23 probe. After 60 seconds, the water level seems to be stationary near the top surface. Due to the high noise, no variation due to the corridor variation is observed, and a flow field visualization could help to find some patterns.

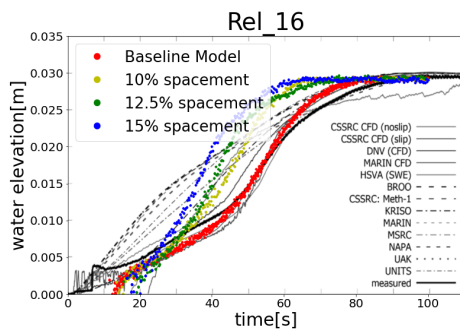


Figure 12: General Arrangement Variation - REL 16

The noise practically vanished in the probe, REL 16, presented in Figure 12, and the first influence on the general arrangement modification is observed. There is a consistent decrease in the time to fill the room as the corridor becomes larger. Another interesting aspect observed is that the time to start the room decreases with the increase in corridor width. It takes almost 20 seconds for the flow to get into the room in the 15% case, and by the high inclination curve, it is possible to infer that the flow moves with higher velocity as the width increases.

REL8 monitor (Figure 13) showed a similar behavior than in the REL16. The time to fill the room was smaller for the wider corridor. However, in contrast to the last monitor, the time to start to fill the room

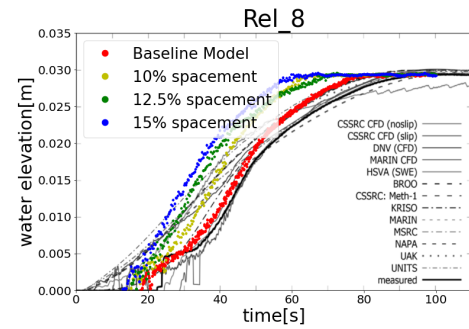


Figure 13: General Arrangement Variation - REL 8

decreased as the corridor width became larger. This resulted in a smaller filling velocity, which can be seen by the curve slopes, which look very similar for the four cases tested.

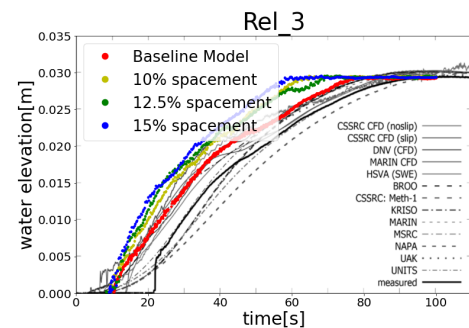


Figure 14: General Arrangement Variation - REL 3

Finally, the probe furthest from the breach, REL3 (Figure 14), showed a slight decrease in the time to fill as the corridor width increased and the time to start filling the room was kept practically constant. Consequently, the velocity to fill the room was slightly different for each case, being faster for the larger spacing and slower for the smaller.

OPEN SOFTWARE FOR AN APPLICATION IN ACADEMY AND INDUSTRY

This paper presented a strong open-source and free CFD tool that revolutionizes the academic and commercial world in terms of engineering research. The idea of the work was to show some of the functionalities of a complex hydrodynamic study case as the flooding problem. The results showed that a parametric study is possible and with reliable results if the user makes a good uncertainty study, chooses the better assumptions and simplifications for the case, and uses good data to compare results.

Potential to Use Openfoam

Of course, in the present paper, not all the capabilities of OF were presented. If we consider just the study case, a lot can be done, such as analyzing flow fields, assessing the transient momentum in the ship, which

would lead to a capsizing, and simulating the ship with more degrees of freedom to capture stability curves. Each complexity added to the case will reflect on the computational cost, which is not linear in most cases. If we extrapolate OF to cases other than the study case, an infinite number of fluid- and thermal-related topics can be studied.

Strong Points

Experimental studies are, in general, very expensive. The costs of a basin tank start from \$3000,00 a day, and there is a necessity to build models using expensive motion capture cameras or more complex methods of flow visualization, such as particle image velocimetry (PIV). The CFD is an ally with a relatively low cost for big sets of cases or parametric studies, as the study case here presented. A highly accurate flow visualization is also an important role of CFD to complement experimental studies.

References

- Georgios Atzampos, Dracos Vassalos, Jakub Cichowicz, Donald Paterson, and Evangelos Boulougouris. esafe-cruise ship survivability in waves. 2019.
- Awel. A brief history of cfd. <https://www.amwel.com/history.html>, 2008. Accessed: 2024-01-19.
- Goong Chen, Qingang Xiong, Philip J Morris, Eric G Paterson, Alexey Sergeev, and Y Wang. Open-foam for computational fluid dynamics. *Notices of the AMS*, 61(4):354–363, 2014.
- Anthony S Daniels and Michael G Parsons. A hybrid agent—genetic algorithm approach to general arrangements. *Ship Technology Research*, 55(2):78–86, 2008.
- Léonard de Vinci. *Del moto e misura dell’acqua*. 1828.
- L Eça and M Hoekstra. Evaluation of numerical error estimation based on grid refinement studies with the method of the manufactured solutions. *Computers & Fluids*, 38(8):1580–1591, 2009.
- Nikolaos D Katopodes. Volume of fluid method. *Free-Surface Flow*, pages 766–802, 2019.
- Gyeong Joong Lee. Dynamic orifice flow model and compartment models for flooding simulation of a damaged ship. *Ocean Engineering*, 109:635–653, 2015.
- Gyeong Joong Lee, Dongkon Lee, Jin Choi, and Hee Jin Kang. A concept study on design alternatives for minimizing accident consequences in maritime autonomous surface ships. *Journal of Marine Science and Engineering*, 11(5):907, 2023.
- Pierre Gilles Lemarié-Rieusset. *The Navier-Stokes problem in the 21st century*. CRC press, 2018.
- Pedro Paludetto Silva de Paula Lopes. *A CFD investigation on the flow around a low aspect ratio vertical cylinder: modeling free surface and turbulent effects*. PhD thesis, Universidade de São Paulo, 2019.
- Reviel Netz. The place of archimedes in world history. *Interdisciplinary Science Reviews*, 47(3-4):301–330, 2022.
- Apostolos Papanikolaou, Byung Suk Lee, Christian Mains, Odd Olufsen, Dracos Vassalos, and George Zaraphonitis. Goals—goal based ship stability & safety standards. *Procedia-Social and Behavioral Sciences*, 48:449–463, 2012.
- Michael G Parsons, Hyun Chung, Eleanor Nick, Anthony Daniels, Su Liu, and Jignesh Patel. Intelligent ship arrangements: a new approach to general arrangement. *Naval Engineers Journal*, 120(3):51–65, 2008.
- Pekka Ruponen, Rinnert van Basten Batemburg, Henry Bandringa, Shuxia Bu, Hendrik Dankowski, Gyeong Joong Lee, Francesco Mauro, Alistair Murphy, Gennaro Rosano, Eivind Ruth, et al. Benchmark study on simulation of flooding progression. In *1st International Conference on the Stability and Safety of Ships and Ocean Vehicles*, 2021.
- Pekka Ruponen, Rinnert van Basten Batenburg, Ri-aan van’t Veer, Luca Braidotti, Shuxia Bu, Hendrik Dankowski, Gyeong Joong Lee, Francesco Mauro, Eivind Ruth, and Markus Tompuri. International benchmark study on numerical simulation of flooding and motions of a damaged cruise ship. *Applied Ocean Research*, 129:103403, 2022.
- Dimitris Spanos and Apostolos Papanikolaou. Safe-dor international benchmark study on numerical simulation methods for the prediction of parametric rolling of ships in wave. *NTUA-SDL Report, Rev, 4*, 2009.
- J Spouge and R Skjong. Risk level and acceptance criteria for passenger ships. *First interim report, part, 2*:1–1, 2014.
- Yu Wang, Hamn-Ching Chen, Arjen Koop, and Guilherme Vaz. Verification and validation of cfd simulations for semi-submersible floating offshore wind turbine under pitch free-decay motion. *Ocean Engineering*, 242:109993, 2021.
- Everett P Wheeler. International conference on safety of life at sea. *American Journal of International Law*, 8(4):758–768, 1914.

AUTHOR BIOGRAPHIES

Pedro P. Lopes holds a Master of Science degree in Naval Architecture and Ocean Engineering from the University of São Paulo, where his research focused on CFD investigations of the flow around low aspect ratio vertical cylinders. Developed a keen interest in the nuclear industry and its unique challenges. He also completed a Bachelor's in Naval Architecture from the Federal University of Santa Catarina, which provided a solid foundation in engineering principles and concepts of naval architecture. Currently a Ph.D. candidate at the University of São Paulo in Severe Accident in NPP simulations. www.linkedin.com/in/pedro-paludetto/

Henrique Gaspar is professor at the Department of Ocean Operations and Technology (NTNU) and active at the Ship Design and Operation Lab. He coordinates the SEUS - Smart European Shipbuilding EU project (<http://seus-project.eu/>). Education consists of a PhD degree in Marine Engineering at the NTNU, with research collaboration at UCL (UK) and MIT (USA). Previous professional experience as a Senior Consultant at Det Norske Veritas (Norway) and in Oil & Gas in Brazil. <https://www.ntnu.edu/ihb/ship-lab>.

Marcos Maturana Naval Architect and Doctor of Science, he is a Researcher at LabRisco-USP, where is currently developing work on risk analysis in the maritime, offshore and nuclear sectors. His work in progress includes the topics of Human Reliability Analysis (HRA) in restricted waters navigation, human performance in control rooms of remotely operated systems, simulation of severe accidents in nuclear power plants, including Floating Nuclear Power Plants (FNPP), Risk-Based Inspection (RBI) of subsea equipment, and decision making in the insurance sector.

Prof. Marcelo Ramos Martins is Professor of Reliability and Risk Analysis in Naval Architecture and Ocean Engineering Graduate Program at the University of São Paulo, since 2001. Visiting Professor in the Reliability Engineering Graduate Program at University of Maryland, USA, between 2010 and 2011 and in the Garrick Risk Science Institute at University of California Los Angeles between 2018 and 2019. Coordinator of Program for Development of Human Resources from the Brazilian National Petroleum, Natural Gas and Biofuels Agency. Director and research leader of LabRisco (Analysis, Evaluation and Risk Management Laboratory) at University of São Paulo and project leader in R&D projects supported by industries and research foundations. Expertise: reliability, maintainability and availability of complex systems; human reliability analysis and quantitative risk analysis.

Prof. Kazuo Nishimoto is Professor of Naval

Architecture and Ocean Engineering. With an Engineering Degree from the Polytechnic School of the University of São Paulo, a Master of Science from Yokohama National University, and a Doctorate from the University of Tokyo, his expertise is well-founded. He has served as a Full Professor at USP since 2006, contributing significantly as a researcher and coordinator in various capacities, including the Numerical Offshore Tank Laboratory. His roles highlight his international experience as a visiting associate at MIT and the University of Michigan, as well as a distinguished professor at Yokohama National University. Committed to advancing research and technology transfer, particularly in CO2 abatement, Kazuo's professional journey reflects a dedication to excellence and innovation in his field.