COMPUTATIONAL FLUID DYNAMICS SIMULATION OF SLOSHING INSIDE BEVERAGE CANS ON A ROTARY FILLING MACHINE

Federico Solari
Natalya Lysova
Federico Scano
Roberto Montanari

University of Parma
Department of Engineering and Architecture
Parco Area delle Scienze 181/A,
43124, Parma, Italy

federico.solari@unipr.it

Enrico Bedogni

Krones AG

Böhmerwaldstraße 5,
93073 Neutraubling,
Germany

Gabriele Copelli

Gabriele Copelli Engineering Office

Via Benedetta 75, 43122, Parma,
Italy

KEYWORDS

Sloshing, rotary filler, CFD simulation, volume of fluid, single reference frame, openFoam, Fluent

ABSTRACT

Sloshing is a critical issue in many industrial contexts. In the food industry, it becomes particularly relevant during the filling of beverage cans and bottles with automatic rotary machines, when the uncapped filled containers move to the transfer star wheel, suddenly changing direction of motion and potentially causing the spilling of the product. Deep knowledge of the system behavior and the fluid dynamics in the domain is essential to guarantee the safety and quality of the final products and processing environment. In this study, Computational Fluid Dynamics (CFD) was used to simulate sloshing in beverage cans using two CFD software: commercial ANSYS Fluent and open-source OpenFOAM. Some modeling strategies are explored with the aim of making the simulation more efficient without impacting the results, and an approach for tracking the maximum fluid level in the can is proposed. The modeling methodology was validated by means of an analytical model and by comparing the results calculated by the two software. Finally, operational insights were derived based on the results of a sensitivity analysis carried out by varying the star wheel diameter and the system productivity.

INTRODUCTION

Sloshing, the transient movement of liquids within a confined container, poses significant challenges in various industries. In transportation and maritime contexts, it can lead to loss of control over the vehicle or compromise the stability of the ship due to the movement of large volumes of fluid.

The phenomenon of sloshing, however, also affects the processing activities of the food industry, mainly in the stages following the filling of containers, when they are transferred towards the sealing station. In this case, the undesirable effect is related to the spilling of fluids, which happens when the product overflows from the container due to the major stresses and acceleration experienced during the motion. Spilling leads to loss of product, contamination of the processing environment, and possible deposition of fluids on the outer walls of the container that could compromise the integrity of the product and the sealing process, undermining both the safety and the quality of the final product.

This phenomenon is particularly relevant during the filling of beverage cans, due to the very limited headspace and the high rotation speeds of the transfer star wheel that increase the probability of sloshing. In this case, the occurrence of sloshing and spilling mainly impacts the seaming of the cans.

A deep understanding of the dynamics of the fluid inside the cans as they travel between the filling and sealing stations, as well as a detailed knowledge about the behavior of the free surface, are crucial for correctly defining the productivity, thus the rotational velocity of the machines, reducing testing times, wastes, and quality issues. In this context, numerical simulation can be of significant support, providing the decision-makers with useful insights and data for detailed analyses, while considerably reducing experimental testing.

Over time, analytical models of sloshing have been developed and validated (Ibrahim 2005) regarding mostly simple geometries and standard boundary conditions. To increase the modeling precision and correspondence to the actual conditions, simulation can be a crucial tool (Elahi et al. 2015). For example, simulation can be used to predict the severity of sloshing in relevant applications (Zheng et al. 2020). As always when it comes to simulation results, validation is required, and it can be usually based on the comparison with theoretical or experimental results (Elahi et al. 2015; Guo et al. 2010). To capture in detail the fluid dynamics involved with the sloshing phenomenon, Computational Fluid Dynamics (CFD) plays a key role, as it allows for modeling complex fluid-structure interactions under various conditions, considering realistic container geometries, custom transfer trajectories, as well as complex and even multiphase fluids usually processed in the food industry. CFD has been applied to simulate a huge variety of food industry processes in the last years

(Szpicer et al. 2023; Ian Wilson and John Chew 2023), also thanks to the increase in the computation resources and the presence of numerous simulation software. To perform CFD simulation, indeed, several commercial and open-source software are available. While commercial CFD software offer benefits such as technical support, updates and enhanced usability, open-source tools provide flexibility and free access, albeit requiring more expertise.

Despite the relevance of sloshing in the beverage industry, to the best of the authors' knowledge, no studies in the literature have dealt with this particular issue to date. Indeed, most articles in the literature have focused on the sloshing of fluids in tanks (Liu and Lin 2008) during transportation, sloshing occurrences and effects in the aerospace (Saltari et al. 2021; Tang et al. 2018) and naval applications, and the different methods that could be implemented to mitigate this phenomenon, e.g., introduction of different geometries and characteristics of baffles (Yu et al. 2020; Zhang 2019). In the context of sloshing in the field of automatic machines, Guagliumi et al. (2021) have presented a technique for analyzing sloshing in cylindrical containers performing rectilinear movements. Guagliumi et al. (2022) have proposed a discrete linear model of sloshing, implementable in real-time for the feedforward anti-sloshing control of container motion.

This article presents a simulation approach, implemented with both commercial ANSYS Fluent and open-source OpenFOAM CFD software, to predict sloshing during the transfer of beverage containers after filling. The calculated inclination angle of the free surface is validated with theoretical data. In addition, using two different tools allows to inter-validate the simulation models by comparing the respective results. Some modeling strategies are explored, aiming at making the simulations more rapid and efficient. Then, the results of the simulations are discussed, proposing an approach for tracking the level of the fluid during the transfer of the container and detecting the occurrence of sloshing. Finally, a sensitivity analysis is performed to assess the effect of different transfer star wheel diameters and productivity levels on the maximum level reached by the fluid.

Table 1: Characteristics of the two carousels

	Diameter [mm]	Number of heads	Rotational speed [rev min^{-1}]
Filling carousel	1080	24	13.89
Transfer star wheel	720	16	20.83

METHODS

Analyzed scenario

The fluid behavior inside a can during the transition from the filling carousel to the transfer star wheel, as depicted in Figure 1, was analyzed. For this purpose, the period from the end of the filling process to the exit from the transfer star wheel was simulated. In the simulated machine, the filling is completed about 40 degrees before the carousel change; it was assumed that, at that time, the free surface conformation within the can was already stabilized.

Since the filling carousel has a larger diameter than the transfer star wheel, and since the tangential velocity must be the same, the latter has a higher rotational speed.

A real industrial context was considered, characterized by a productivity of 20'000 cans per hour (c/h). 0.5-litre can geometry, having an inner diameter of 0.033 m and height of 0.15 m, was modeled.

Table 1 reports the geometric characteristics and the operating conditions assumed for the two carousels. Based on the data, the law of motion of the can, in terms of the evolution of angular velocity over time, and as a consequence, of centrifugal acceleration over time, can be derived.

In Figure 2, the angular velocity and the centrifugal acceleration characterizing the motion of the can are presented: at 0.5 s the can is assumed to transfer from the filler to the star wheel.

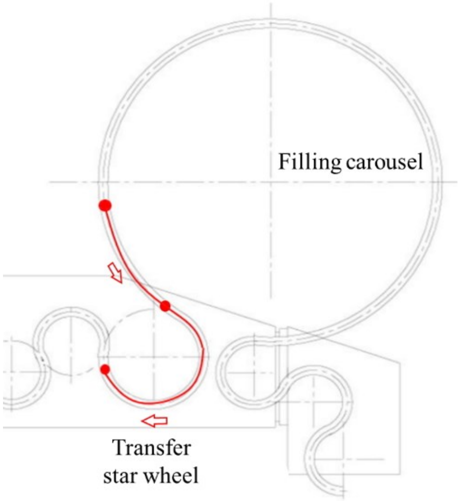

Figure 1: Scheme of the simulated line

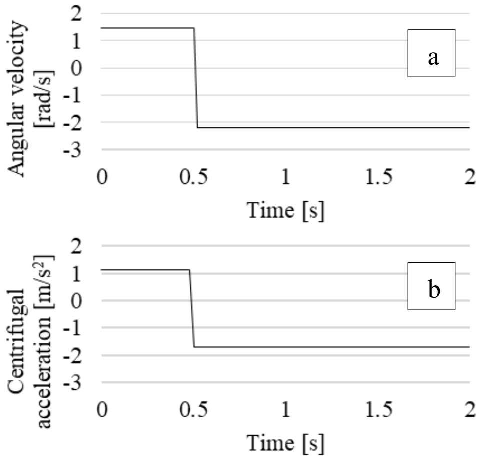

Figure 2: a) angular velocity and b) centrifugal acceleration characterizing the motion of the can in the simulated system

Table 2: Characteristics of the fluids

	Air	Water
Density [kg m^{-3}]	1.225	998.2
Dynamic viscosity [Pa s]	1.8e-5	1.0e-3
Surface tension [N m^{-1}]	0.072	

Numerical simulation

In this study, we explore the capabilities of CFD simulation for studying sloshing phenomena in industrial bottling processes.

Aiming to assess the reliability of the proposed approach, the results obtained with two different software, namely ANSYS Fluent and OpenFOAM, were compared.

The single reference frame (SRF) method was adopted to reproduce the behavior of the fluid inside the can as it moved within an absolute reference frame with a given law of motion. This method consists of an alternative approach, less computationally demanding compared to the traditional moving mesh method, that can be adopted when simulations involve moving regions, allowing to study them in their respective reference frames.

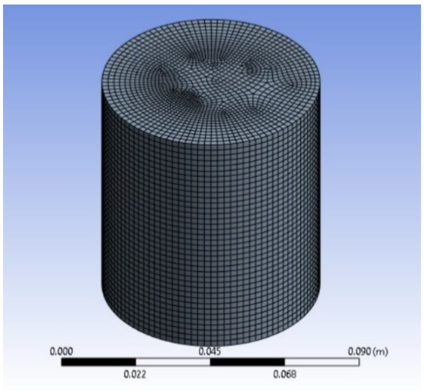

Figure 3: Hexahedral mesh of the computational domain

A multiphase simulation was set up using the Volume-of-Fluid (VOF) method, which is appropriate for the simulation of immiscible fluids being separated by a well-defined interface surface (free surface) (Hirt and Nichols, 1981). Air and water were considered as fluids, whose properties, although well known, are summarized in Table 2.

Under the conditions considered, a significant but not abrupt free surface perturbation was expected, not affecting the entire volume of fluid contained in the can, but only the upper part. For this reason, it was decided to include only the upper 3 cm of the can in the calculation, as the lower part of the cylinder contributed minimally to the sloshing dynamics. This assumption was validated by comparing the results obtained by simulating the whole can geometry with those obtained with the reduced domain.

To ensure a high-resolution capture of the fluid behavior, a hexahedral mesh, with equilateral elements having a dimension of 3 mm, was defined (Figure 3). To better compare the results obtained, the same mesh was used with both solvers.

The law of motion was finally assigned to the mesh by imposing the acceleration field represented in Figure 2.

Moreover, a gravitational field, with a downward acceleration of 9.81 m/s^2, was applied.

Inclination of the free surface

To identify the initial condition, i.e., the shape of the liquid-free surface inside the can at the end of the filling process, especially in terms of inclination angle (θ) with respect to the horizontal plane, an ad-hoc simulation was conducted. Starting from a resting condition (θ=0°), the revolution of the can on the filling carousel was simulated until the stationarity condition was reached. The inclination of the free surface was then measured and compared with the angle calculated by solving the balance of forces acting on the fluid, as described also in Elahi et al. (2015):

$$\theta = arc\ tan\left(\frac{a_r}{g}\right) \qquad (1)$$

Where a_r is the centrifugal acceleration in the radial direction and g is the gravitational acceleration, acting in the y direction.

This comprehensive procedure served as a robust foundation for determining the initial free surface angle in the first carousel and ensured accurate initialization for subsequent simulations. Furthermore, it allowed to validate the simulation approach with well-established theoretical notions.

After the initial inclination angle was obtained and validated as described, the simulation of the transition phase from the filling carousel to the transfer star wheel was performed. The simulation was run using both software packages, setting the angle determined as the initial condition. The simulation outcomes were then compared, to check whether the results obtained with both software, in terms of liquid sloshing inside the can, starting from an identical initial condition, were the same. This further step is intended as an inter-validation step of obtained results.

Maximum fluid level Ymax

To qualitatively assess sloshing, it is sufficient to observe the behavior of the free surface during the movement of the can. To make a more accurate and precise assessment, the maximum height reached by the free surface during the can motion, *Ymax*, was also calculated and traced over time. This parameter can be very useful in assessing the impact sloshing may have on the process quality and efficiency. When *Ymax* is lower than the total height of the can, even if sloshing is happening, the liquid doesn't leak out. In this scenario, therefore, sloshing does not represent an issue. On the other hand, when *Ymax* exceeds the maximum height of the can, the liquid may overflow from the can and negatively affect the filling process.

The comparison of the results obtained with the two software was then performed in two ways:

- a comparison of the trend over time of the maximum height reached by the free surface.
- a frame-by-frame comparison between the shape of the free surface computed by the two software in specific time intervals.

Finally, the proposed method was used to evaluate the effect that some operational and design parameters have on liquid behavior. In particular, the impact of transfer star wheel diameter and line productivity was evaluated.

RESULTS

The first phase of the study aimed to identify the computational domain. To have a trade-off between computational time and results accuracy, it was decided to compare the results achieved considering only the upper part of the can (30 mm), with the results obtained considering the whole can. The results are shown in Figure 5a.

Based on the graph, it can be observed that as the can moves from the filling carousel to the transfer star wheel (t=0.5 s) sloshing occurs. A wave is generated that reaches its maximum peak 0.14 s after the transfer and then continues fluctuating throughout the period considered. The results of the two simulations are perfectly overlapping: the percentage deviation between the two curves is always less than 3.9%. It can therefore be concluded that the liquid in the bottom of the can does not significantly affect the dynamics of sloshing and can consequently be excluded from the computational domain.

The second phase of the study focused on identifying the liquid condition at the exit of the filling carousel. To validate this condition, the angle of inclination of the free surface under the effect of the centrifugal force generated by the rotation of the carousel was calculated in two different ways. The balance of forces acting on the free surface (eq. 1) leads to an angle value of 6.64 degrees. Below are the results of a simulation that, starting from a resting condition (θ=0°), simulates the revolution of the can on the filling carousel until the inclination angle of the free surface is stabilized (Figure 4).

Measuring the inclination angle that the free surface reaches under stationary conditions yields a theta value of 6.24 degrees which results in a deviation of 6% from the theoretical value, calculated with eq.1. This deviation can be considered acceptable because it is included within the range of accuracy that the mesh size allows for.

The following phase of the study focused on the sloshing occurring as the beverage can moved from the filling carousel to the transfer star wheel. A comparison between the results obtained with ANSYS Fluent and those obtained with OpenFOAM, is shown in Figure 5b for the value of *Ymax*, while a representation of the free surface, calculated with the two software, at several time steps is presented in Figure 6.

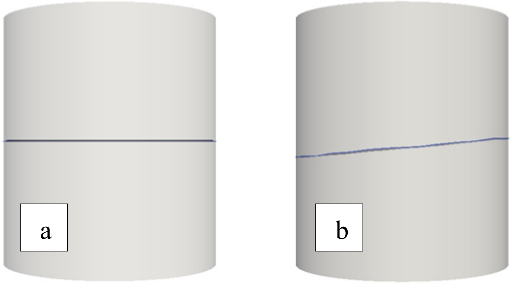

Figure 4: Initial (a) and final (b) state of the free surface

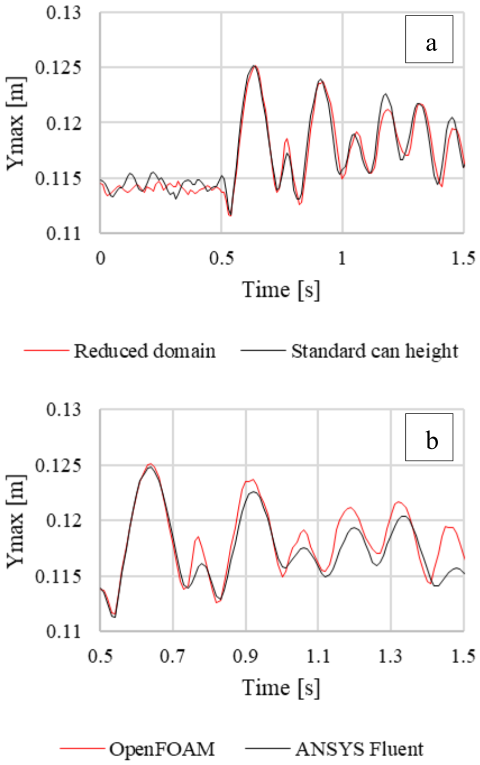

Figure 5: a) comparison of standard vs reduced domain; b) comparison of results calculated with OpenFOAM vs ANSYS Fluent

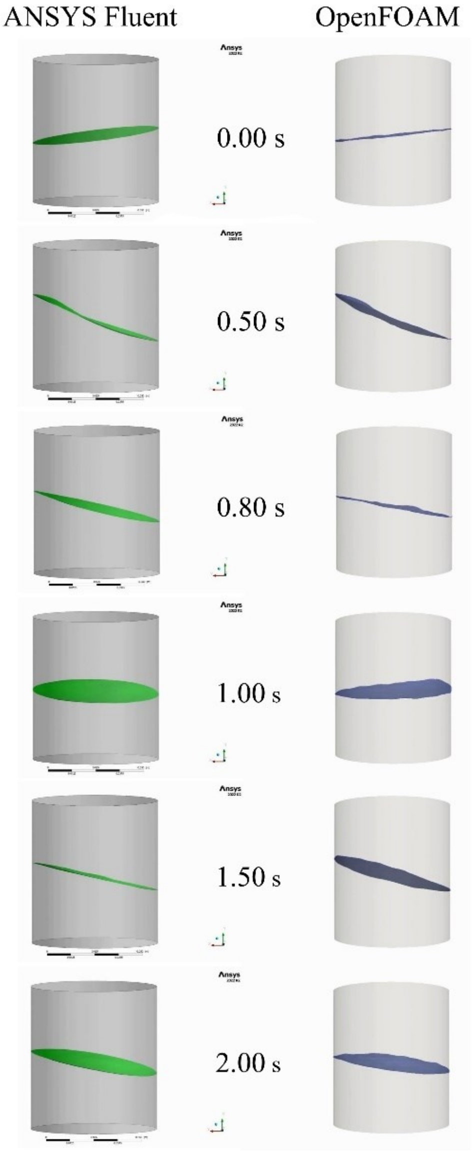

Figure 6: Free surface of the fluid inside the can at different time steps, calculated with ANSYS Fluent (left) and OpenFOAM (right)

It can be concluded that the results of the two CFD software agree both in the prediction of the maximum fluid level reached inside the can and in the oscillation frequency of the wave itself. Even the shapes of the free surface calculated by the two software appear to be in good agreement.

Finally, the proposed simulation approach was used to evaluate the impact that some processing and geometric parameters have on liquid behavior. The evolution of *Ymax* reached by the liquid inside the can for different diameters of the transfer star wheel is presented in Figure 7a, while the levels observed with different productivity values are shown in Figure 6
7b.

It can be seen that the maximum height reached by the liquid decreases as the diameter of the transfer star increases.

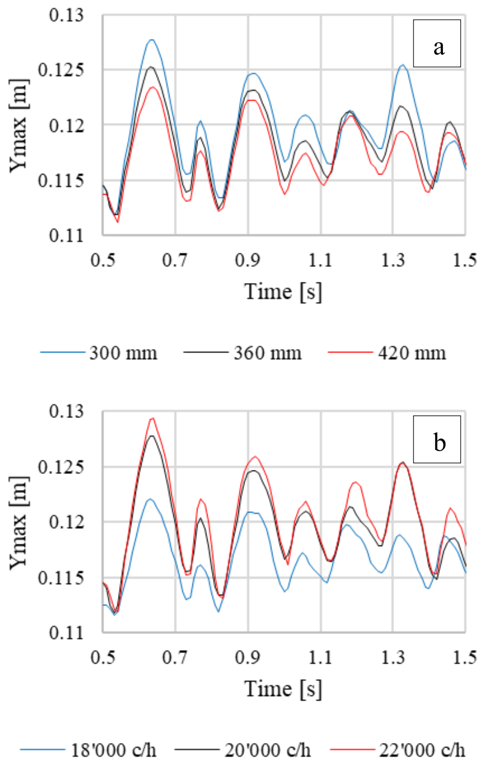

Figure 7: *Ymax* with a) different star wheel diameters and b) different productivity levels

This correlation appears to be linear. On the other hand, regarding the correlation between *Ymax* and productivity, it can be seen that between 18’000 and 20’000 cans per hour, there is a much greater difference than between 20’000 and 22’000 cans per hour. This indicates that this correlation is probably not linear in nature. This aspect needs to be investigated in more detail in future research studies.

CONCLUSIONS

In the present study, a method for predicting liquid sloshing occurring during container motion in transfer carousels during the filling process is presented.

In-depth knowledge of this phenomenon is a big advantage in both the design and management phases of automatic filling. A rigorous step-by-step approach was followed to identify the optimal simulation settings that would allow to obtain accurate results in times consistent with industrial needs. The reliability of the simulation approach was first validated by comparing the results with those obtained from an analytical model in a simple case study (i.e., can in uniform rotary motion around a fixed axis).

A subsequent inter-validation phase, conducted by comparing the results obtained with two different software (i.e., ANSYS Fluent and OpenFOAM) in a more complex case study, reproducing a real-world context in which the can passes from the filling carousel to the transfer star, confirmed the consistency of the results. The findings suggest that both tools can significantly support the optimization phase of the industrial processes by accurately modeling complex fluid dynamics, thereby reducing testing time and enhancing production efficiency.

The developed approach represents a novelty within the scientific literature because to date there is no study outlining fluid dynamic simulation models to simulate sloshing during filling processes. The presented approach allows to evaluate the behavior of the liquid inside the can between the end of the filling process and the seaming by allowing to assess the impact that some design and operational parameters have on the behavior of the liquid. In future activities, experimental tests will be performed to further validate the proposed approach.

REFERENCES

Elahi, R., Passandideh-Fard, M., & Javanshir, A. (2015). Simulation of liquid sloshing in 2D containers using the volume of fluid method. Ocean Engineering, 96, 226–244. https://doi.org/10.1016/j.oceaneng.2014.12.022

Guo, L. C., Zhang, S., Morita, K., & Fukuda, K. (2010). Fundamental validation of the finite volume particle method for 3D sloshing dynamics. International Journal for Numerical Methods in Fluids, 68(1), 1–17. Portico. https://doi.org/10.1002/fld.2490

Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics, 39(1), 201–225. https://doi.org/10.1016/0021-9991(81)90145-5

Guagliumi, L., Berti, A., Monti, E., & Carricato, M. (2022). Antisloshing Trajectories for High-Acceleration Motions in Automatic Machines. Journal of Dynamic Systems, Measurement, and Control, 144(7). https://doi.org/10.1115/1.4054224

Guagliumi, L., Berti, A., Monti, E., & Carricato, M. (2021). A Simple Model-Based Method for Sloshing Estimation in Liquid Transfer in Automatic Machines. IEEE Access, 9, 129347–129357. https://doi.org/10.1109/access.2021.3113956

Ian Wilson, D., & John Chew, Y. M. (2023). Fluid mechanics in food engineering. Current Opinion in Food Science, 51, 101038. https://doi.org/10.1016/j.cofs.2023.101038

Ibrahim, R. A. (2005). Liquid Sloshing Dynamics. https://doi.org/10.1017/cbo9780511536656

Liu, D., & Lin, P. (2008). A numerical study of three-dimensional liquid sloshing in tanks. Journal of Computational Physics, 227(8), 3921–3939. https://doi.org/10.1016/j.jcp.2007.12.006

Saltari, F., Traini, A., Gambioli, F., & Mastroddi, F. (2021). A linearized reduced-order model approach for sloshing to be used for aerospace design. Aerospace Science and Technology, 108, 106369. https://doi.org/10.1016/j.ast.2020.106369

Szpicer, A., Bińkowska, W., Wojtasik-Kalinowska, I., Salih, S. M., & Półtorak, A. (2023). Application of computational fluid dynamics simulations in food industry. European Food Research and Technology, 249(6), 1411–1430. https://doi.org/10.1007/s00217-023-04231-y

Tang, Y., Yue, B., & Yan, Y. (2018). Improved method for implementing contact angle condition in simulation of liquid sloshing under microgravity. International Journal for Numerical Methods in Fluids, 89(4–5), 123–142. Portico. https://doi.org/10.1002/fld.4685

Yu, L., Xue, M.-A., & Zhu, A. (2020). Numerical Investigation of Sloshing in Rectangular Tank with Permeable Baffle. Journal of Marine Science and Engineering, 8(9), 671. https://doi.org/10.3390/jmse8090671

Zhang, E. (2019). Numerical research on sloshing of free oil liquid surface based on different baffle shapes in rectangular fuel tank. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, 234(2–3), 363–377. https://doi.org/10.1177/0954407019855569

Zheng, M.-Z., Gou, Y., Teng, B., & Jo, H. (2020). A practical prescreening method for sloshing severity evaluation. Petroleum Science, 17(4), 1119–1134. https://doi.org/10.1007/s12182-020-00453-x

AUTHOR BIOGRAPHIES

FEDERICO SOLARI is a researcher and lecturer at the Department of Engineering and Architecture of the University of Parma since September 2021. He graduated (with distinction) in Mechanical Engineering for the Food Industry in 2008 and got his Ph.D. in Industrial Engineering in 2014, both at the University of Parma. He authored 39 publications indexed on Scopus. His main research topics are industrial plant logistics, industrial plant analysis and design, supply chain management, advanced industrial plant design also using simulative techniques. In his research activities, he explores the use of simulation and advanced numerical techniques for the design, control and maintenance of industrial plants and processes. His e-mail address is: federico.solari@unipr.it and his Web- page can be found at ttps://personale.unipr.it/it/ugovdocenti/person/102724.
ORCID: 0000-0001-8365-965X

NATALYA LYSOVA is a Ph.D. Candidate at the University of Parma, where she graduated in Engineering for the Food Industry in 2021. Her doctoral research project is titled “Virtualization approaches for industrial plants control and design”. In her research activities, she aims to leverage

numerical techniques for the analysis, design and optimization of industrial systems, plants, and devices. Her e-mail address is: natalya.lysova@unipr.it.
ORCID: 0000-0001-6066-6398

FEDERICO SCANO is a Master's student at the University of Parma. He is now concluding his studies in Engineering for Food Industry and this work will be part of his dissertation.

ROBERTO MONTANARI is a Full Professor at the Department of Engineering and Architecture - University of Parma. He is the Holder of the course in Industrial Plants – Bachelor of Engineering Management, and the course in Simulation of Production Systems – Master Degree in Engineering for the Food Industry. His e-mail address is: roberto.montanari@unipr.it and his Web- page can be found at https://personale.unipr.it/it/ugovdocenti/person/19786.

ENRICO BEDOGNI is an R&D Mechanical Engineer. He got his Ph.D. in Industrial Engineering in 2013 at the University of Parma. After a few years as a researcher at the University of Parma, he spent several years in the R&D field, in Automatic Machines and Automotive sectors. He is now leading the Krones Innovation Hub. His e-mail address is: enrico.bedogni@krones.com

GABRIELE COPELLI is a Mechanical Engineer; from 2005 to 2013 research fellow at the Department of Industrial Engineering of the University of Parma, in the machinery area. Freelance since 2013. He is expert in mechanical design and development of machinery and processes for the food industry by means of CAE (Computer Aided Engineering) and Computational Fluid Dynamics (CFD). www.gabrielecopelli.com