

Computational Fluid Dynamics (CFD) Analysis of Fuel Sloshing in a Tanker Truck with Different Design of Baffles

Arif Mohd Fauzi¹, Norrizal Mustaffa^{1*}

¹ Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia, Parit Raja, 86400, MALAYSIA

*Corresponding Author: cd200199@student.uthm.edu.my
DOI: <https://doi.org/10.30880/rpmme.2024.05.02.016>

Article Info

Received: 4 July 2024

Accepted: 14 October 2024

Available online: 31 December 2024

Keywords

Fuel Sloshing, Baffle Design, Tanker Truck, Computational Fluid Dynamics (CFD), ANSYS Fluent, Wall Shear Stress, Force Analysis,

Abstract

This study presents a comprehensive analysis of the effectiveness of different baffle designs in mitigating fuel sloshing within tanker trucks using computational fluid dynamics (CFD) simulations with ANSYS Fluent. The primary objective was to compare two distinct baffle designs and evaluate their performance across various motion phases: acceleration, constant velocity and deceleration. The results demonstrate that Baffle Design 2 consistently outperforms Baffle Design 1 in controlling fuel movement, reducing turbulence, and minimizing wall shear stress. During the acceleration and deceleration phases, Baffle Design 1 exhibited significant fuel displacement and turbulence, indicating higher instability. In contrast, Baffle Design 2 maintained a more controlled and localized fuel distribution, reflecting superior ability to mitigate fuel movement and reduce sloshing. Force analysis further revealed that Baffle Design 1 experienced higher and more variable forces, suggesting greater instability and dynamic impact on the tank structure. Conversely, Baffle Design 2 exhibited lower and more consistent forces, indicating better control of sloshing and reduced dynamic loads on the tanker. The findings underscore the critical importance of optimized baffle designs in enhancing the safety and durability of tanker trucks. Baffle Design 2's superior performance in mitigating fuel sloshing not only ensures better fuel stability but also contributes to the overall structural integrity of the tanker. These insights are crucial for designing safer and more efficient fuel transportation systems. The study provides valuable guidelines for future baffle design development aimed at reducing operational risks and improving tanker truck longevity.

1. Introduction

One of the primary issues encountered in fuel transportation is fuel sloshing. Sloshing refers to the movement of the liquid cargo inside the tank, which can lead to several adverse effects. Fluid sloshing has a negative impact on the roll dynamics and braking properties of such vehicles [1]. Fluid sloshing occurs when the vehicle accelerates, decelerates, or travels on uneven roads, causing the liquid inside to move and create sloshing forces.

Sloshing has been studied by using the Computational Fluid Dynamics (CFD) method [2-9]. To reduce the effects of fuel sloshing, the use of baffles within the tanker is a common practice [10-17]. Baffles are internal structures that obstruct the free movement of the liquid, thus reducing the amplitude and energy of the sloshing waves. Various baffle designs have been proposed and implemented, each with different levels of effectiveness.

However, there remains a need for a comprehensive analysis of how different baffle designs impact fuel sloshing under various operating conditions.

Despite the critical nature of this issue, there is lack of specific regulations or standardized guidelines governing the design and implementation of baffles within tanker trucks to mitigate sloshing effects. Existing baffle designs vary widely, and their effectiveness in reducing fuel sloshing during acceleration and deceleration remains inadequately understood.

Therefore, this study aims to evaluate the fuel sloshing behavior in a tanker truck with different baffle designs using computational fluid dynamics (CFD) simulations. The research focuses on comparing the performance of these baffle designs to determine the optimal configuration that minimizes fuel sloshing. Previous studies have demonstrated that CFD analysis is effective for conducting detailed investigations into the hydrodynamics of fluid sloshing within tank structures. This technique is essential for observing and analyzing the effects of different baffle designs on fuel movement.

2. Methodology

2.1 Geometry Modelling of the Tank and Baffles

The size of the tanks used in this study are according to the specifications given by Panda Mechanical Co. Ltd [18]. Computer-Aided Design (CAD) software, SolidWorks was used to create all the tanks and baffles geometry. For the simulation, CFD Software, Ansys Fluent was used. Figure 1 shows the geometry drawing which included the length, width and height of the tank used for the simulation.

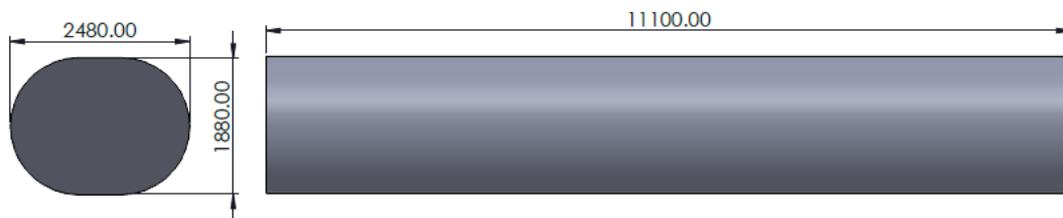


Fig. 1 Geometry model of the Tank

Figure 2 shows Baffle Design 1 and Baffle Design 2. The baffles are placed every 2000 mm along the tank's length.

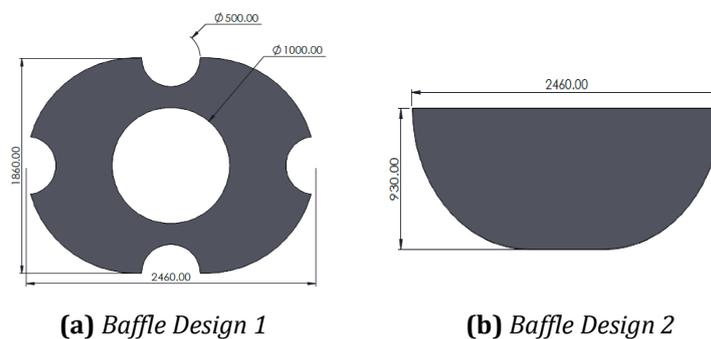
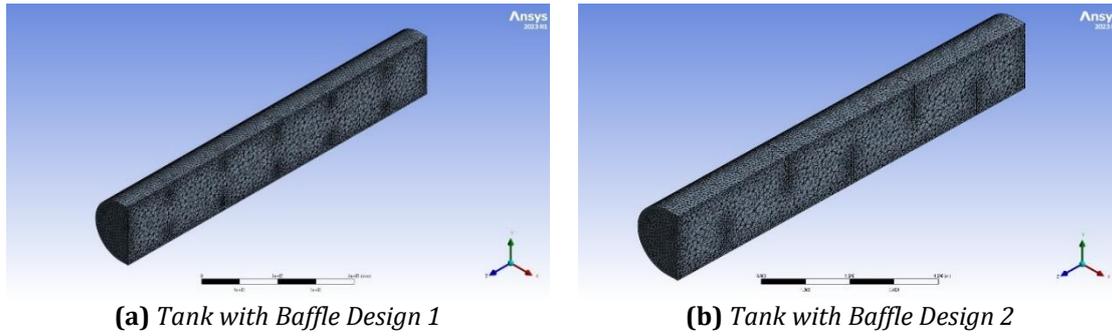


Fig. 2 Geometry model of the Baffles (a) Baffle Design 1; (b) Baffle Design 2

2.2 Mesh Generation

Meshing is a crucial step before conducting numerical simulations, as it breaks down complex geometries into simpler elements that can be used for local approximation. Generating an excessively large number of cells during meshing can lead to prolonged and inefficient computational times. Creating too few cells can result in inaccurate simulation outcomes. Hence, the choice of meshing method is vital, as it directly impacts the accuracy, convergence, and speed of the simulation.

Figure 3(a) illustrates the mesh generation for tank with the baffle design 1 while Figure 3(b) shows the mesh generation for tank with baffle design 2. The tetrahedron approach was used to mesh the model due to its simplicity.



(a) Tank with Baffle Design 1 **(b) Tank with Baffle Design 2**
Fig. 3 Mesh Generation (a) Tank with Baffle Design 1; (b) Tank with Baffle Design 2

2.3 Governing Equation

The Navier-Stokes equations describe the motion of viscous fluid substances. These equations are a set of nonlinear partial differential equations that account for the conservation of mass, momentum, and energy in the fluid.

2.3.1 Continuity Equation

The Continuity Equation is written as,

$$\nabla \cdot u = 0 \tag{1}$$

2.3.2 Momentum Equation

The Momentum Equation is written as,

$$\text{x-momentum: } \rho \frac{Du}{Dt} = \frac{\partial(-p+\tau_{xx})}{\partial x} + \frac{\partial\tau_{yx}}{\partial y} + \frac{\partial\tau_{zx}}{\partial z} + S_{Mx} \tag{2}$$

$$\text{y-momentum: } \rho \frac{Dv}{Dt} = \frac{\partial\tau_{xy}}{\partial x} + \frac{\partial(-p+\tau_{yy})}{\partial y} + \frac{\partial\tau_{zy}}{\partial z} + S_{My} \tag{3}$$

$$\text{z-momentum: } \rho \frac{Dw}{Dt} = \frac{\partial\tau_{xz}}{\partial x} + \frac{\partial\tau_{yz}}{\partial y} + \frac{\partial(-p+\tau_{zz})}{\partial z} + S_{Mz} \tag{4}$$

2.3.3 Turbulence Modelling

For this study, standard **k-ε model** had been chosen as the selected turbulence model. The turbulence kinetic energy, **k**, and its rate of dissipation, **ε**, are obtained from the following transport equation.

$$\frac{\partial}{\partial t}(pk) + \frac{\partial}{\partial x_i}(pk u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b + Y_M + S_k \tag{5}$$

$$\frac{\partial}{\partial t}(p\varepsilon) + \frac{\partial}{\partial x_i}(p\varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \tag{6}$$

$$C_{1\varepsilon} = 1.44, \quad C_{2\varepsilon} = 1.92, \quad C_\mu = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3$$

2.4 Fluid Properties

The physical properties of fluids are important for predicting the sloshing in the tank. Table 1 shows the fluid properties used for the simulation.

Table 1 Fluid Properties

Properties	Air	Diesel-liquid
Density (kg/m ³)	1.255	730
Viscosity (kg/m.s)	1.7894×10 ⁻⁵	0.0024
Surface Tension Coefficient	-	0.025

2.5 Simulation Setup

The CFD simulation setup using ANSYS Fluent. include the solver type, multiphase model configuration, fluid properties, and the turbulence model employed. These parameters are critical for accurately simulating the fuel sloshing within the tanker truck, ensuring that the results are both realistic and reliable. Table 2 shows the solver settings and Turbulence model used in the simulation.

Table 2 Solver Setting

Type	Pressure-Based
Velocity Formulation	Absolute
Time	Transient
Gravity Acceleration	-9.81 (y-direction)
Volume of Fluid	Phase 1: Air, Phase 2: Diesel-Liquid
Viscous Model	k-epsilon, standard
Solution Method	
Scheme	PISO
Gradient	Least Squares Cell Based
Pressure	Second Order Upwind
Momentum	Second Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Turbulent Dissipation Rate	First Order Upwind

3. Result and Discussions

3.1 Force Analysis

The graph in Figure 4 shows the force exerted on the tank walls over time for two different baffle designs. Both baffle design 1 and baffle design 2 exhibit fluctuating force patterns over the 12 second interval, indicative of dynamic sloshing behavior within the tanker. For baffle design 1, the force fluctuates significantly, with notable peaks and troughs throughout the time, suggesting substantial impact from the fluid movement. In comparison, baffle design 2 also exhibits significant force fluctuations, but generally with a lower magnitude compared to baffle design 1. The maximum force for this design is lower than the baffle design 1, indicating potentially better mitigation of sloshing forces. These results underscore the importance of baffle design in managing the dynamic forces within tanker trucks, with baffle design 2 potentially offering better protection against sloshing-induced forces

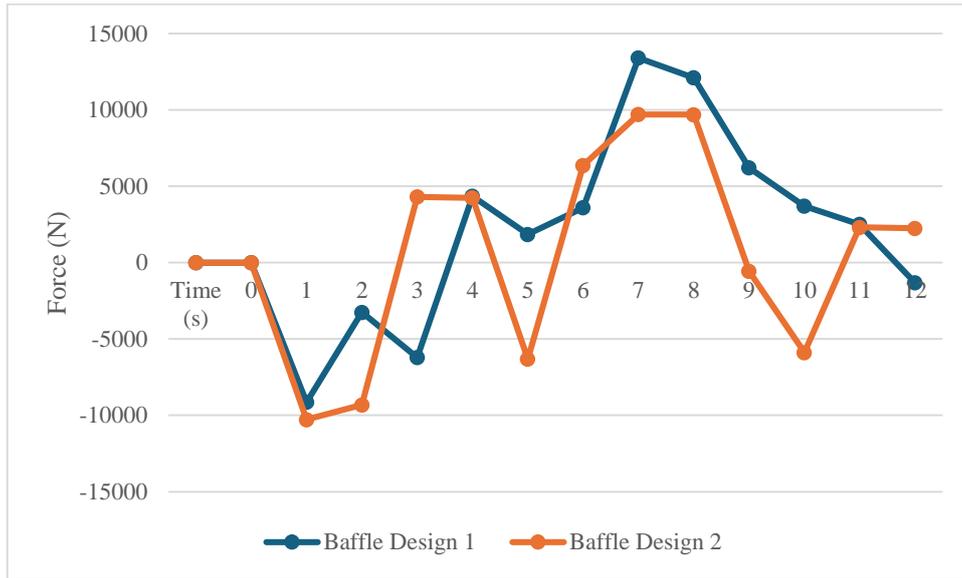


Fig. 4 Graph of Force Analysis

3.2 Stress Analysis

The graph in Figure 5 illustrates the stress distribution over time for the two different baffle designs in the tanker truck. Both baffle designs exhibit fluctuating stress levels over the 12-second interval, indicative of the sloshing. Comparing the two baffle designs, baffle design 1 experiences higher peak stress levels compared to baffle design 2, indicating that the first design may be less effective at mitigating stress induced by sloshing. The higher stress peaks in baffle design 1 could lead to greater wear and potential structural damage over time. On the other hand, baffle design 2 shows a more stable and lower stress distribution, suggesting better performance in reducing the impact of sloshing. The lower and more consistent stress levels in baffle design 2 indicate improved structural integrity and potentially longer service life for the tank.

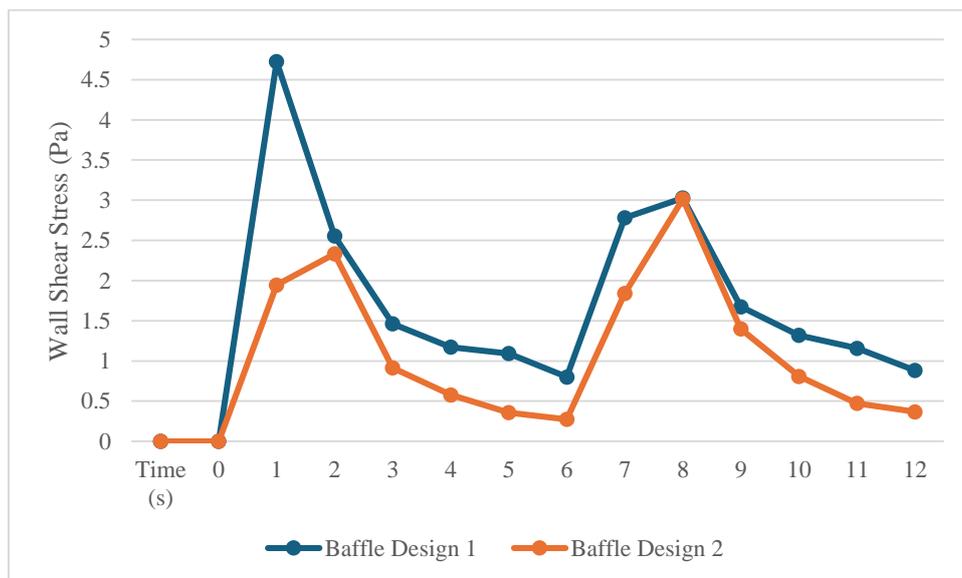


Fig. 5 Graph of Wall Shear Stress Analysis

3.3 Contour of Volume of Fluid (VOF) and Wall Shear Stress

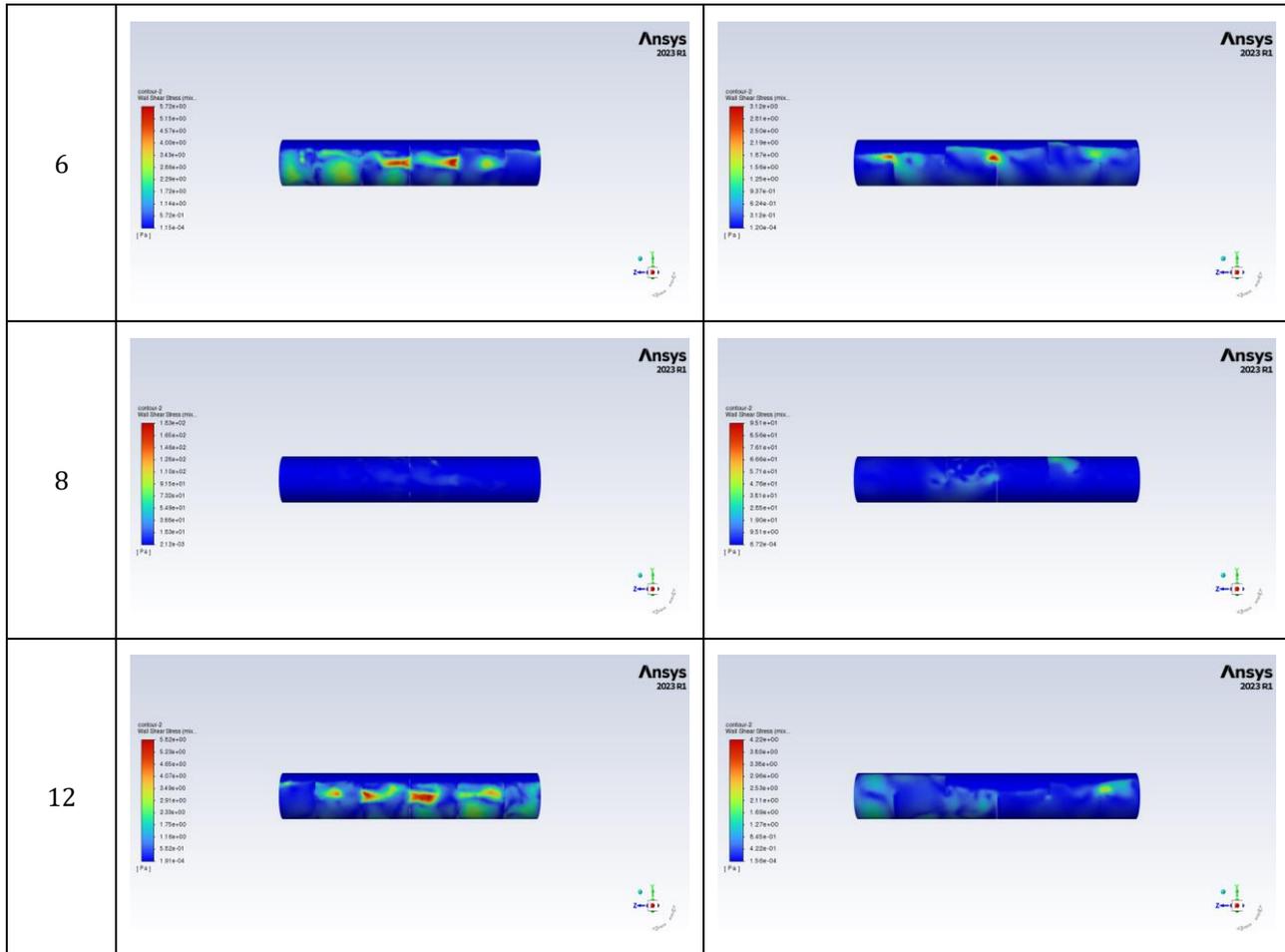
Table 3 shows the contour of VOF while Table 4 shows the contour of Wall shear Stress.

Table 3 Contour of VOF

Time (s)	Tank with Baffle Design 1	Tank with Baffle Design 2
2		
6		
8		
12		

Table 4 Contour of Wall Shear Stress

Time (s)	Tank with Baffle Design 1	Tank with Baffle Design 2
2		



The simulation results provide a comprehensive assessment of the performance of two different baffle designs in mitigating fuel sloshing within a tanker truck under various motion conditions. This discussion synthesizes the key findings from the VOF distribution and wall shear stress analyses, highlighting the comparative effectiveness of baffle design 1 and baffle design 2. The VOF distribution analysis reveals that baffle design 2 performs better compared to baffle design 1 in controlling fuel sloshing across all motion phases. During the acceleration and deceleration phase, tank with baffle design 1 exhibits significant fuel displacement and turbulence, indicating a higher degree of instability. In tank with the baffle design 2 demonstrates a more controlled and localized fuel distribution, reflecting its superior ability to mitigate fuel movement and reduce sloshing. Overall, the findings indicate that Baffle Design 2 consistently outperforms Baffle Design 1 in mitigating fuel sloshing across all motion phases. Baffle Design 2 demonstrates superior control over fuel movement, reduced turbulence, and lower wall shear stress, resulting in improved stability and reduced sloshing effects. The force and stress analyses further support these conclusions, highlighting the importance of effective baffle design in enhancing the safety and structural integrity of tanker trucks by minimizing the dynamic impacts of fuel sloshing. The comprehensive analysis underscores the need for optimizing baffle designs to ensure the safe and efficient transport of liquid fuels in tanker trucks.

4. Conclusion

This study provides a thorough comparative analysis of two distinct baffle designs in mitigating fuel sloshing within a tanker truck, using CFD simulations. The primary objective was to evaluate the effectiveness of these baffle designs under various motion conditions, including acceleration, constant velocity and deceleration. The results unequivocally demonstrate that Baffle Design 2 consistently outperforms Baffle Design 1 across all motion phases. Baffle Design 2 shows superior control over fuel movement, reduced turbulence, and lower wall shear stress, resulting in improved stability and minimized sloshing effects. During the acceleration and deceleration phases, Baffle Design 1 exhibits significant fuel displacement and higher instability, whereas Baffle Design 2 maintains a more controlled and localized fuel distribution. The insights gained from this research provide valuable guidelines for the future development of baffle designs. They emphasize the role of effective baffle configurations in reducing operational risks and improving the longevity of tanker trucks. This study serves as a foundational work that underscores the significance of advanced CFD simulations in optimizing industrial designs

for safety and efficiency. It sets the stage for further research aimed at refining baffle designs and exploring new materials and configurations to achieve even better performance in fuel sloshing mitigation.

Acknowledgement

The author would like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support.

References

- [1] T. Kandasamy, "An Analysis of Baffles Designs for Limiting Fluid Slosh in Partly Filled Tank Truck," *The Open Transportation Journal*, vol. 4, no. 1, pp. 23–32, 2010, doi: 10.2174/1874447801004010023.
- [2] R. Ambade and R. Kale, "CFD Analysis of Sloshing Within Tank," *International Journal of Innovative Research in Technology & Science*, vol. 2, no. 3, p. 6, 2014.
- [3] Z. Cai, A. Topa, L. P. Djukic, M. T. Herath, and G. M. K. Pearce, "Evaluation of Rigid Body Force in Liquid Sloshing Problems of a Partially Filled Tank: Traditional CFD/SPH/ALE Comparative Study," *Ocean Engineering*, vol. 236, no. July, 2021, doi: 10.1016/j.oceaneng.2021.109556.
- [4] E. Frosina *et al.*, "Study of the Sloshing in a Fuel Tank using CFD and EFD Approaches," *ASME/BATH 2017 Symposium on Fluid Power and Motion Control, FPMC 2017*, no. September 2018, 2017, doi: 10.1115/FPMC2017-4337.
- [5] C. D. Chaudhari, "Analysis Fluid Sloshing when Road Tanker Experience Sudden Breaking-A computational approach," *International Journal of Engineering and Applied Physics (IJEAP)*, vol. 1, no. 2, pp. 154–161, 2021, [Online]. Available: <https://ijeap.org/>
- [6] G. S. Brar and S. Singh, "An Experimental and CFD Analysis of Sloshing in a Tanker," *Procedia Technology*, vol. 14, pp. 490–496, 2014, doi: 10.1016/j.protcy.2014.08.062.
- [7] Y. Kang, Z. Wu, S. Ma, M. Zhao, and W. Li, "CFD: Based Assessment and Visualization of the Failure Consequences of LPG Tankers," *J Loss Prev Process Ind*, vol. 82, no. August 2022, p. 105008, 2023, doi: 10.1016/j.jlp.2023.105008.
- [8] V. Singal, J. Bajaj, N. Awalgaonkar, and S. Tibdewal, "CFD Analysis of a Kerosene Fuel Tank to Reduce Liquid Sloshing," *Procedia Eng*, vol. 69, pp. 1365–1371, 2014, doi: 10.1016/j.proeng.2014.03.130.
- [9] G. B. Micheli, M. L. F. Fogal, V. L. Scalon, A. Padilha, and K. A. R. Ismail, "Analysis and Reduction of the Sloshing Phenomena Due To Sudden Movement of Spray Mixture Tanker," vol. 15, no. 2, pp. 399–413, 2022.
- [10] R. Thundil Karuppa Raj, T. Bageerathan, and G. Edison, "Design of Fuel Tank Baffles to Reduce Kinetic Energy Produced by Fuel Sloshing and to Enhance the Product lLife Cycle," *ARNP Journal of Engineering and Applied Sciences*, vol. 9, no. 3, pp. 244–249, 2014.
- [11] W. Siraye, A. Wakjira, and R. B. Nallamothe, "An Analysis of Fuel Oil Sloshing in Partially Filled Cargo Tanker Trucks under Cornering Conditions Using Various Baffle Systems," *Journal of Engineering (United Kingdom)*, vol. 2023, 2023, doi: 10.1155/2023/9941864.
- [12] J. L. Bautista-Jacobo, E. Rodríguez-Morales, J. J. Montes-Rodríguez, and H. Gámez-Cuatzín, "Effect of Baffles on the Sloshing in Road Tankers Carrying LPG: A Comparative Numerical Study," *Math Probl Eng*, vol. 2015, 2015, doi: 10.1155/2015/359470.
- [13] C. A. Bellezi, L. Y. Cheng, and K. Nishimoto, "A Numerical Study on Sloshing Mitigation by Vertical Floating Rigid Baffle," *J Fluids Struct*, vol. 109, p. 103456, 2022, doi: 10.1016/j.jfluidstructs.2021.103456.
- [14] P. Huang, "A Comprehensive Investigation on Liquid Sloshing of Rectangular Water Tank with Vertical Baffles," *Ocean Engineering*, vol. 288, no. P2, p. 116126, 2023, doi: 10.1016/j.oceaneng.2023.116126.
- [15] X. L. Zheng, X. S. Li, Y. Y. Ren, Y. N. Wang, and J. Ma, "Effects of Transverse Baffle Design on Reducing Liquid Sloshing in Partially Filled Tank Vehicles," *Math Probl Eng*, vol. 2013, 2013, doi: 10.1155/2013/130570.
- [16] U. O. Ünal, G. Bilici, and H. Akyıldız, "Liquid Sloshing in a Two-dimensional Rectangular Tank: A Numerical Investigation with a T-shaped Baffle," *Ocean Engineering*, vol. 187, no. November 2018, 2019, doi: 10.1016/j.oceaneng.2019.106183.
- [17] N. Ahmad, M. Varshney, and M. H. Farooqi, "Design and CFD Analysis of Baffles of Fuel Tanker Trucks for Normal and Grade Highway Conditions," *International Journal of Forensic Engineering and Management*, vol. 1, no. 1, p. 76, 2020, doi: 10.1504/ijfem.2020.109178.
- [18] Panda Mechanical Co Ltd., "Liquid Fuel Tank Semi Truck Trailer." Accessed: Dec. 25, 2023. [Online]. Available: <https://www.pandamech.com/category/tank-trailer>