CFD Study of an EV Under Wading Condition Using Simcenter STAR-CCM+

Tiago Augusto Santiago Vieira
Davi Machado Cury
Pedro Henrique Araujo
Aline Amaral Quintella Abdu
Caiuã Caldeira de Melo
Fábio de Castro Orefice
Henrique Carlos Monteiro
Siemens Digital Industries Software

ABSTRACT

In response to the increasing demand for the performance of Electric Vehicles (EVs), there is a growing interest in examining the effects of challenging conditions, such as wading through water, on these vehicles. Given that most previous studies on wading have focused on conventional Internal Combustion Engine Vehicles (ICEVs), automotive manufacturers are endeavoring to enhance their understanding of wading in the context of EVs. While wading is primarily assessed through physical testing, applying Computational Fluid Dynamics (CFD) offers a more thorough analysis and can significantly reduce costs and time. The present work presents a methodology for study wading through CFD simulations. The CFD code Simcenter STAR-CCM+ (2410) was employed to perform the simulations. This code was used for geometry preparation, mesh generation, solving, and post-processing the achieved results. A trimmed mesh was generated for the fixed parts of the EV and the wheels. The numerical approach employs the Volume of Fluids (VOF) method for multiphase modeling and Reynolds Average Navier-Stokes (RANS) for turbulence modeling. This study concentrates on the topology of the water surface and the forces exerted on the vehicle. The forces applied by the water pressure on the EV surfaces are evaluated since they can provide valuable insights into the sealing of the cabin and the battery. The results show pressure spots on the bottom of the EV and the water velocity profile through time. In conclusion, the use of CFD simulations, specifically employing the presented methodology, proves to be highly beneficial in evaluating and optimizing the water tightness of the EV to prevent issues regarding the EV electric motor and battery.

INTRODUCTION

The capability of vehicles to navigate through water, referred to as wading, is a crucial performance criterion for automotive manufacturers. Ensuring that vehicles can traverse water safely and effectively without compromising their functionality or safety is of paramount importance. Computational Fluid Dynamics (CFD) has emerged as a powerful tool for simulating and analyzing the complex interactions between water and vehicle components during wading scenarios [1, 2].

Automotive wading simulations are essential for evaluating vehicle performance when driving through water at various depths and speeds. These simulations enable engineers to understand the impact of water on different vehicle components, such as the engine, transmission, and electrical systems. The primary objective is to ensure that vehicles can traverse water safely without compromising their functionality or safety. This involves testing the integrity of components like the bumper, engine undertray, and radiator [1].

To conduct these simulations, several strategies are available in the literature. Primarily, there are moving or static vehicle setups. The moving setup mimics real-world conditions where the vehicle drives through a water channel, offering a more realistic but computationally intensive approach. In contrast, the static setup keeps the vehicle stationary while simulating water flow around it, which is less computationally demanding but may not capture dynamic fluid behaviors as effectively. Another approach is to simplify the setup before full vehicle tests, such as using simplified block tests. These involve using a rectangular block at different speeds and immersion depths to validate the simulation method, helping to verify the basic principles of fluid dynamics and interaction with solid objects [3].

Automotive wading simulations face several challenges due to the complex interactions between the vehicle and water. One of the primary difficulties is accurately modeling the interaction between water and vehicle components. The dynamic nature of water, including splashes and waves, requires sophisticated simulation techniques to capture these effects accurately. This complexity is further increased by the need to simulate the fluid-structure interaction (FSI) in real-time, ensuring that the vehicle's response to water is realistically portrayed [4].

Another significant challenge is simulating the impact of water on various vehicle components, such as the engine undertray and electrical systems. These components must be tested for durability under different water depths and speeds to ensure they can withstand real-world conditions. Durability testing involves assessing how water exposure affects the materials and functionality of these components over time. This aspect of simulation is crucial for ensuring the reliability and safety of vehicles in wading conditions [5].

Real-world wading conditions can vary widely, adding another layer of complexity to the simulations. Factors such as water temperature, debris, and varying water depths must be accurately represented to create realistic scenarios. These environmental variables can significantly influence the vehicle's performance and durability. Therefore, simulating these variable conditions accurately is essential for developing vehicles that can handle diverse and unpredictable wading environments [4, 5].

Beyond traditional wading simulations, this issue becomes even more important when analyzing Electrical Vehicles (EVs). The wading process is particularly crucial for EVs due to their reliance on electrical systems and components that must remain functional and safe even when exposed to water. Unlike traditional internal combustion engine vehicles, EVs have high-voltage batteries, electric motors, and complex electronic control units that are sensitive to water ingress. Ensuring that these components are well-protected during wading scenarios is essential to prevent short circuits, electrical failures, and potential safety hazards. Effective wading simulations help engineers design robust sealing and waterproofing measures, ensuring that EVs can safely navigate through water without compromising their electrical integrity.

Moreover, the importance of the wading process for EVs extends to their overall reliability and consumer confidence. As EV adoption increases, consumers expect these vehicles to perform reliably in diverse environmental conditions, including heavy rain and flooded roads. Demonstrating that EVs can handle such conditions without issues is vital for building trust in this technology. Comprehensive wading simulations and testing provide valuable data that can be used to improve the design and durability of EVs, ultimately contributing to their market acceptance and long-term success. By addressing the unique challenges posed by water exposure, manufacturers can ensure that EVs are not only environmentally friendly but also resilient and dependable in real-world scenarios [6].

This manuscript presents a comprehensive study on the application of CFD techniques to automotive EVs wading simulations. The primary objectives are to evaluate the fluid-structure interactions and assess the impact on vehicle components. By leveraging advanced CFD methods, such as the Volume of Fluid (VOF) method for multiphase modeling and Reynolds-Averaged Navier-Stokes (RANS) for turbulence modeling, this research aims to provide insights into the dynamic behavior of water around the vehicle and the resultant forces exerted on critical components. The CFD code Simcenter STAR-CCM+ (version 2410) was employed to perform the simulations [7]. The developed methodology utilized a mesh only around the EV, making the setup less computationally expensive. A trimmed mesh was generated for the fixed parts of the EV and the wheels. However, this approach includes simplifications, such as no wheel movement or water tank depth. Through this investigation, we aim to enhance the understanding of automotive wading dynamics and contribute to the development of more robust and reliable vehicles capable of performing under diverse environmental conditions.

METHODOLOGY

In the realm of physics, the wading problem is characterized by the interaction between air and water flows. Initially, the domain comprises a water pit, while the remaining area is filled with air. As the electric vehicle (EV) progresses through the domain, it encounters this water pit, resulting in the redistribution of water and its subsequent interaction with the EV. Typically, the entire domain is meshed, encompassing the EV, air, and water regions. The EV is represented by a moving mesh (overset mesh), while the rest of the domain remains static. To accurately capture the rotation of the wheels, additional overset meshes can be embedded within the EV's moving mesh. This configuration allows for the precise simulation of splashes caused by the wheels and the EV's movement. The mesh schematics for this approach are illustrated in Figure 1. However, this traditional method is computationally intensive, necessitating small time steps and highly refined meshes.

Furthermore, the complexity of the wading problem necessitates a detailed and meticulous approach to meshing. The interaction between the EV and the water pit involves intricate fluid dynamics that require high-resolution meshes to accurately capture the nuances of the flow. The overset mesh technique, while effective in representing the movement of the EV and the resulting splashes, demands significant computational resources. This is due to the need for tight time steps to ensure stability and accuracy in the simulation. Consequently, the classical approach, while robust, is often accompanied by substantial computational costs, making it imperative to balance precision with efficiency in CFD calculations.

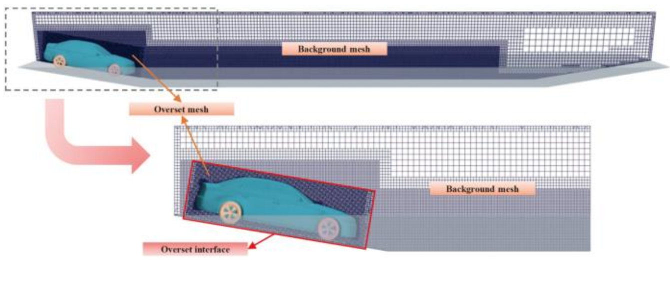

Figure 1 – Classical meshing approach for wading simulations [3].

To streamline the numerical setup, an alternative approach involves utilizing only the mesh surrounding the electric vehicle (EV) and assuming a vertical threshold for the air and water interface. This configuration is depicted in Figure 2, which illustrates the proposed geometric and mesh setup, highlighting the air-water interface. According to this setup, the domain above the threshold is considered as air, while the domain below is water. This method results in a significantly smaller mesh and allows for larger time steps, as the domain of interest is confined to the vicinity of the EV.

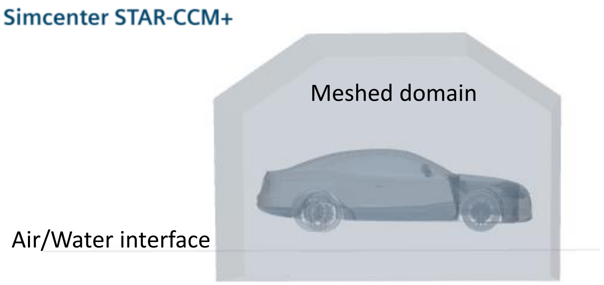

Figure 2 – Geometrical and mesh setup of the current work.

However, there are two notable limitations to this approach. Firstly, the water flow does not interact with the floor and side walls of the actual domain, which is a scenario that occurs in real wading situations. Secondly, the rotation of the wheels is not incorporated in this setup, although it is planned for future studies. Despite these limitations, this approach enables rapid simulations, providing quick insights for engineers to understand the product's behavior. The proposed method can be integrated with the classical approach after product refinements and knowledge-gathering phases. This comprehensive approach will be evaluated in future wading studies to enhance the accuracy and reliability of the simulations.

GEOMETRY AND MESH - A detailed view of the utilized domain, or volume of interest, is illustrated in Figure 2. In contrast, Figure 3 presents the precise geometry of the electric vehicle (EV) employed in the study. The EV in question is a generic sedan, equipped with all necessary structural components to facilitate the wading calculation. This particular EV geometry has been effectively utilized in previous research for paint shop calculations, as referenced in other studies [8, 9]. Furthermore, as previously indicated in Figure 2, the EV is enclosed within a defined volume of interest, which constitutes the actual mesh domain for the simulation.

This comprehensive setup ensures that the EV's structural integrity and interaction with the surrounding fluid dynamics are accurately represented. The inclusion of all essential components within the EV geometry allows for a realistic simulation of the wading scenario, providing valuable insights into the vehicle's performance under such conditions. The defined volume of interest serves as the focal point for the mesh domain, ensuring that the computational resources are efficiently allocated to the areas of greatest significance. This approach not only enhances the accuracy of the simulation but also optimizes the computational efficiency, making it a robust method.

Figure 3 – EV geometry used for mesh generation.

The adopted meshing strategy involved the use of trimmed (Cartesian) meshes. Trimmed meshes, typically composed of hexahedral cells, present numerous advantages for Computational Fluid Dynamics (CFD) calculations. One of the foremost benefits is their capacity to generate high-quality, structured grids that align effectively with the flow direction. This alignment minimizes numerical diffusion and significantly enhances the accuracy of the simulation, particularly in regions exhibiting complex flow patterns. Furthermore, trimmed meshes are adept at managing anisotropic spacing, which facilitates efficient boundary layer resolution and a more precise representation of gradients. This capability leads to faster convergence and more reliable solutions, making trimmed meshes particularly suitable for simulations where precision is paramount [7].

In addition to their alignment with flow direction, trimmed meshes offer substantial computational efficiency. The structured nature of these meshes allows for deterministic indexing, which accelerates the process of determining cell connectivity. This efficiency translates to reduced CPU time and lower computational costs, making trimmed meshes a cost-effective choice for large-scale simulations. Moreover, the ability of trimmed meshes to handle complex geometries without compromising mesh quality ensures that the solver can produce accurate and stable results. Consequently, the use of trimmed meshes in CFD calculations strikes a balance between accuracy, efficiency, and computational resource management, thereby optimizing the overall simulation process.

Another notable advantage of trimmed meshes lies in their computational efficiency. These meshes can be generated relatively swiftly and feature deterministic indexing, which speeds up cell connectivity determination. This efficiency translates into reduced CPU time and lower computational costs, rendering trimmed meshes a cost-effective option for large-scale simulations. Moreover, the structured nature of trimmed meshes ensures that the solver can adeptly handle complex geometries without compromising mesh quality, resulting in more accurate and stable outcomes. Overall, the utilization of trimmed meshes in CFD calculations strikes a balance between accuracy, efficiency, and computational resource management.

The generated mesh and its specifics are illustrated in Figure 4. This figure depicts the volume of interest surrounding the electric vehicle (EV) and the meshed EV itself. As previously mentioned, the entire domain was meshed using a trimmed grid [7]. The generated mesh comprises slightly fewer than 9 million cells, in contrast to the classical approach documented in the literature, where meshes typically contain close to 20 million cells [1, 2, 3, 4, 5]. This significant reduction in cell count not only enhances computational efficiency but also facilitates quicker simulations, providing engineers with rapid insights into the product's behavior. Consequently, the adoption of trimmed meshes offers a pragmatic solution for achieving high-fidelity simulations while managing computational resources effectively.

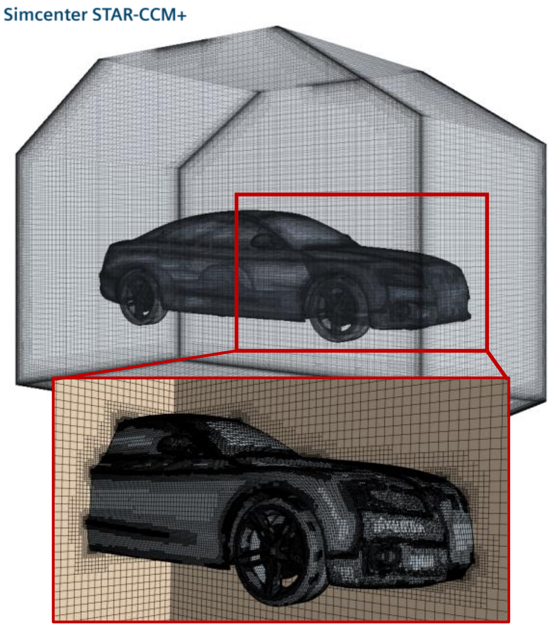

Figure 4 – Meshed volume of interest and EV.

BOUNDARY CONDITIONS - The boundary conditions applied to the generated mesh are detailed in Figure 5. The symmetry condition assumes an infinite water tank on the sides of the domain, which may be an oversimplification. This assumption will be evaluated in future research, where the classical approach will be compared with the methodology presented in this study. The top of the domain is designated as a pressure outlet, allowing air and/or water to move freely through this boundary. The front, rear, and bottom of the domain are modeled as velocity inlets, as they are the initial boundaries to come into contact with the water. The geometry of the electric vehicle (EV) is set as a wall, with mesh refinements near the EV surface to accurately capture the viscous layer flow (refer to Figure 4).

To maintain a constant and convergent Courant number, a time step of 0.01 seconds was used, with a first-order time discretization. The total simulated physical time was 18.5 seconds, sufficient for the EV to exit the water and return above the air-water interface. The simulation employed the Volume of Fluid (VOF) method and k-ε turbulence modeling. This setup ensures that the boundary conditions are appropriately defined to capture the complex interactions between the EV and the surrounding fluid dynamics. Future studies will further refine these conditions to enhance the accuracy and reliability of the simulations, providing deeper insights into the wading behavior of the EV.

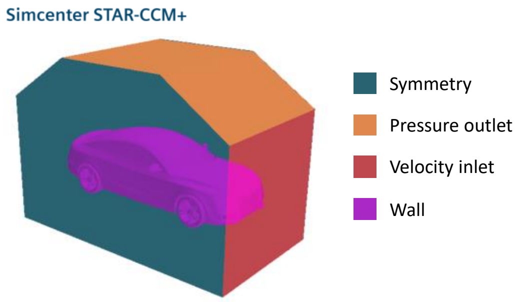

Figure 5 – Employed boundary conditions.

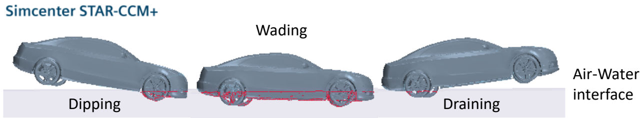

Figure 6 – EV movement and stages during the simulation.

Regarding the movement of the electric vehicle (EV), a velocity of approximately 10 km/h was assumed, which is typical for wading simulations [4, 5]. The EV's movement was prescribed using a CSV file, detailing the water ingress and egress at an angle of 8 degrees. Additionally, the EV penetrated approximately 0.5 meters into the water below the interface, a value consistent with the available literature. These assumed parameters are subject to variation to understand the behavior of the EV and the fluid flow under different conditions. However, the primary objective here is to evaluate the proposed methodology as a proof-of-concept. The stages of the EV's movement during the simulation are depicted in Figure 6.

This approach allows for a detailed analysis of the EV's interaction with the water, providing insights into the dynamics of wading scenarios. By prescribing the movement through a CSV file, the simulation can accurately replicate the ingress and egress of water, ensuring that the EV's behavior is realistically modeled. The penetration depth of 0.5 meters is particularly significant, as it aligns with established research, thereby validating the simulation parameters. Future studies may explore variations in these values to further refine the understanding of the EV's performance in wading conditions, but the current focus is to demonstrate the viability of the proposed methodology.

RESULTS

The results achievable from wading calculations encompass a wide range of information, including pressure fields, water flow around the electric vehicle (EV), and forces acting on specific components. In the current study, the primary focus is on the forces exerted by the fluid-structure interaction (FSI) on the underbody of the EV, the corresponding pressure field in qualitative terms, and the water distribution at different physical times. The forces in Cartesian coordinates are depicted in Figure 7. This figure illustrates that no symmetry was observed in the lateral interactions between the EV and the water, as evidenced by the differing amplitudes of the X and Y force components. This asymmetry suggests the presence of cutting forces applied to the underbody screws and welding spots, which are structurally weaker points.

Conversely, the Z force component exhibited the highest amplitude, which is anticipated given the direction of the EV's movement. The Z force component has the potential to cause deformation of metal sheets beyond their elastic limit, potentially leading to component failure. Therefore, the Z force is a critical parameter to evaluate the product's structural integrity under such conditions. This is particularly crucial for EVs, as it pertains to the protection of the electric motor and battery bank. The analysis of these forces provides valuable insights into the structural behavior of the EV during wading.

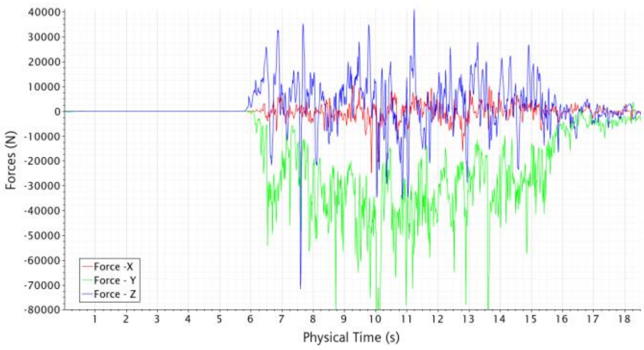

Figure 7 – Forces applied in the EV during the simulation.

In conjunction with the forces depicted in Figure 7, the dimensionless pressure field is illustrated in Figure 8. The decision to present the pressure field qualitatively stems from the wide range of values, which complicates direct comparison. At 5.9 seconds of solution time, Figure 8 shows that the highest-pressure values are concentrated at the front part of the underbody. This behavior is anticipated, as is the moment the EV enters the water tank (air-water interface). As the simulation progresses, the pressure field exhibits a more distributed pattern, as seen at 6.8 seconds of solution time. The qualitative assessment of the pressure field and water distribution further aids in understanding the dynamic interactions at play, contributing to the overall evaluation of the EV's performance in wading scenarios.

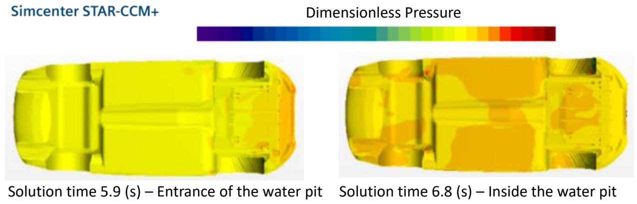

Figure 8 – Forces applied in the EV during the simulation.

The pressure field enables the identification of the most critical areas, and correlating these results with the force amplitudes allows for the assessment of the durability and quality of the EV components. This is particularly significant because the battery is typically located on the underbody, precisely where the highest forces are encountered in wading situations. The welded and screwed sheets in this region are subjected to substantial forces, which can impact the structural integrity of the EV.

Figure 9 illustrates the water distribution on the electric vehicle (EV) during the wading process. At 5.9 seconds, only the front portion of the underbody is in contact with the water pit. This observation aligns with the onset of higher amplitude forces (refer to Figure 7) and the pressure distribution depicted in Figure 8. As time progresses, the water predominantly accumulates at the front part of the EV, as evidenced at 6.8 seconds. At this juncture, the effects of model simplifications, such as the absence of wheel movement, become apparent. According to the literature, when wheel movement is included, the splash zone extends to encompass the EV doors and the rear part of the underbody, which is not observed in this case.

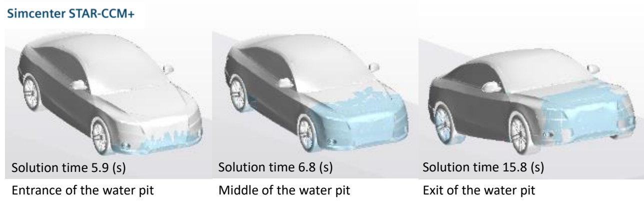

Figure 9 – Water distribution during the simulation.

Nevertheless, the front part of the EV should still be safeguarded from water, as it is subject to significant splashing, which persists until 15.8 seconds. At this point, the water body on top of the hood begins to drain as the EV exits the air-water interface. This behavior underscores the importance of protecting the front section of the EV from water ingress, as it is the most affected area during the wading process. The analysis of water distribution, in conjunction with force and pressure data, provides a comprehensive understanding of the EV's performance under wading conditions, highlighting critical areas that require attention to ensure the vehicle's durability and safety.

Regarding the computational cost associated with the presented methodology and simulation, around 16h were required as the total elapsed time required to complete the calculation. The computational resource utilized was a 40-core HPC single node, which is somewhat excessive given that Simcenter STAR-CCM+ recommends approximately 250,000 cells per core for optimal parallelization. Nevertheless, the achieved simulation time demonstrates that with this computational setup, it is feasible to obtain high-quality results and insights in less than a day. This timeframe allows for multiple iterations of the product before proceeding to production.

Moreover, this rapid turnaround enables the acquisition of valuable insights, which can then be applied to a classical wading approach for final verification and refinement. The ability to perform several iterations within a short period is crucial for optimizing the design and ensuring that the product meets all necessary specifications. By leveraging the computational power of a 40-core HPC node, engineers can efficiently explore various scenarios and make informed decisions, ultimately enhancing the overall quality and reliability of the final product. This approach underscores the importance of balancing computational resources with the need for thorough and iterative analysis in the development process.

CONCLUSIONS

This manuscript presents an extensive study on the application of CFD techniques to simulate wading scenarios for automotive EVs. The primary objectives are to evaluate fluid-structure interactions and assess their impact on vehicle components. By leveraging advanced CFD methods, such as the VOF method for multiphase modeling and RANS for turbulence modeling, this research aims to provide insights into the dynamic behavior of water around the vehicle and the resultant forces exerted on critical components. The simulations were performed using the CFD code Simcenter STAR-CCM+ (version 2410) [7]. The methodology developed used a mesh exclusively around the EV, making the setup less computationally expensive. A trimmed mesh was generated for the fixed parts of the EV and the wheels. However, this approach includes simplifications, such as the absence of wheel movement and water tank depth. Through this investigation, we aim to enhance the understanding of automotive wading dynamics and contribute to the development of more robust and reliable vehicles capable of performing under diverse environmental conditions.

The developed model yielded results regarding the forces applied during the wading situation, which are consistent with the pressure field observed during different phases of the EV's entry and movement within the water pit. The water distribution on the EV during the simulation was concentrated on the hood, indicating the simplification of no wheel movement. This aspect will be addressed in future studies. The presented methodology proved to be a valuable asset, as it enabled the rapid acquisition of detailed information about the EV. This knowledge provides valuable insights for modifying and enhancing the product before production. Employing this methodology allows for multiple product iterations, thereby improving its quality, reliability, and durability. In a subsequent step of the proposed methodology, a classical wading simulation could be conducted to ensure that the assumptions for the final product are accurate.

For future research, the goal will be to compare the proposed methodology to the same case with wheel movement. Additionally, a comparison with the classical wading approach will be made to assess computational time gains and the real impact of the proposed simplifications. Comparisons with Eulerian Multiphase instead of VOF for multiphase solving will also be performed. Furthermore, a relatively new feature of Simcenter STAR-CCM+, Smoothed Particle Hydrodynamics, which is a meshless approach to the same problem, will be evaluated.

REFERENCES

1. Khapane, Prashant, and Uday Ganeshwade. *Wading simulation-challenges and solutions*. No. 2014-01-0936. SAE Technical Paper, 2014.
2. Zheng, X., X. Qiao, and F. Kong. "Vehicle Wading Simulation with STAR-CCM+: Proceedings of the FISITA 2012 World Automotive Congress." 2013.
3. Zhang, Huashan, et al. "3D large-scale SPH modeling of vehicle wading with GPU acceleration." *Science China Physics, Mechanics & Astronomy*. Vol 66. 2023.
4. Olsson, Elin. "CFD Modeling of Wading with Electric Vehicles. Development of a Simulation Method Using OpenFOAM." Master thesis, Chalmers University of Technology. 2018.
5. Varnousfaderani, Shayan Rahat. "Numerical Computation of Vehicle Wading. A Study Using the VOF Method." Master thesis, Chalmers University of Technology. 2013.
6. Melo, Caiuã Caldeira, et al. *Battery Development for Vehicles: Integration of 3D Finite Element Method and CFD for Effective Thermal Management*. No. 2024-36-0163. SAE Technical Paper, 2024.
7. Siemens Digital Industries Software. Simcenter STAR-CCM+, version 2023.2, Siemens 2023.
8. Vieira, Tiago Augusto Santiago, et al. *CFD Analysis of the Paint Shop Curing Process.* SIMEA, 2024
9. Vieira, Tiago Augusto Santiago, et al. *A Computational Fluid Dynamics Methodology to Predict Automotive Painting Process Using Simcenter STAR-CCM+*. No. 2023-36-0056. SAE Technical Paper, 2024.