Advancements in Marine Hydrodynamics: Insights from the Black Sea Region through Computational Fluid Dynamics

Vlad George PANAIT (1.), Cosmin BERESCU (2.)

1. Universitatea Politehnica din București, Romania
2. Universitatea Maritimă Constanța, Romania

Abstract:
This comprehensive article delves into the latest research on marine hydrodynamics within the Black Sea, particularly focusing on areas around Midia Cape-Constanta and the Danube river discharge through the Sulina branch. It integrates findings from three distinct studies, each employing computational fluid dynamics (CFD) to analyze marine current regimes, the interaction of structures with fluid flows, and hydrological phenomena. Key insights include the assessment of navigation risks, understanding the impact of structures like ships on fluid flows, and a comparative analysis of flow phenomena prediction models. This synthesis offers a holistic view of the current state of marine hydrodynamics research in this region, highlighting both the capabilities and limitations of CFD in understanding and predicting complex aquatic systems.

Introduction

The first study delves into the hydrodynamic behavior around the Midia Cape-Constanta zone. It focuses on the interaction of riverine and marine currents, which creates a challenging environment for navigation and marine constructions. Using advanced CFD models, the study analyzes the seasonal variations in current patterns and their impact on sediment transport and erosion. This research is essential for planning and maintaining safe navigation channels and for understanding the environmental impact on the coastline.

The second study investigates the Sulina branch of the Danube River, emphasizing the unique hydrodynamic conditions created by the river's discharge into the Black Sea. This study utilizes CFD to model the complex interplay between the river flow and the sea's tidal forces. The findings highlight the implications for sediment deposition, which affects both navigational safety and ecological balance in the delta region. The study also provides valuable insights for managing the river's outflow to minimize adverse environmental effects.

The final study focuses on the design and optimization of hydro-technical constructions in the Black Sea region. This research applies CFD to simulate the interaction of marine structures with the dynamic sea environment. By analyzing wave patterns, current dynamics, and sediment transport, the study offers critical guidance for the construction of ports, breakwaters, and other marine infrastructure. The goal is to ensure these structures are both efficient and environmentally sustainable, minimizing their impact on the surrounding marine ecosystem.

In conclusion, the combined findings from these three studies offer a comprehensive understanding of the marine hydrodynamics in the Black Sea region. The use of CFD as a research tool provides a detailed and accurate analysis of the complex interactions between marine and riverine environments. These insights are invaluable for informed decision-making in navigation, construction, and environmental management in this strategically important area.

Study 1: Assessing Marine Current Regime in Midia Cape-Constanta Area

The first study under consideration provides a detailed analysis of the marine current regime near Midia Cape-Constanta. Utilizing CFD, this research assesses the distribution of velocities and pressures based on local hydro-meteorological characteristics. It offers vital insights into identifying risk areas for navigation and the existing hydro-technical construction industry. The modeling results showcase the current velocities, pressures, and turbulences in the marine region. This model is further expanded to analyze coastal changes and assess risks for hydraulic structures. The incorporation of sea wave characteristics into the model provides an estimate of the potential hydro-dynamic power at specific locations.

Importance of Assessing Marine Current Regime in Midia Cape-Constanta Area

The Midia Cape-Constanta area is known for its rich marine biodiversity and strategic location. Assessing the marine current regime in this area is of utmost importance for various reasons. Firstly, it helps us understand the flow and direction of water, which is crucial for safe navigation and shipping activities. By knowing the currents, we can plan routes effectively and avoid potential hazards.

Secondly, assessing the marine current regime is essential for fisheries management. Fishermen rely on the knowledge of currents to determine the best locations for fishing, optimizing their catch and ensuring the sustainability of marine resources. Understanding the currents also helps in predicting the movement patterns of fish and other marine organisms, aiding in their conservation efforts.

Lastly, the assessment of marine currents is vital for coastal infrastructure management. Constanta is a major port city, and understanding the currents is crucial for the design and maintenance of harbors and other coastal structures. By considering the impact of currents, engineers can ensure the stability and longevity of these structures, protecting them from erosion and other potential risks.

Methods and Tools Used for Assessing Marine Current Regime

Assessing the marine current regime requires advanced technology and scientific expertise. Several methods and tools are commonly used to gather data and analyze the currents in the Midia Cape-Constanta area.

One of the primary tools used is Acoustic Doppler Current Profilers (ADCP). ADCPs measure water velocity and direction using sound waves, providing detailed information about the currents. These devices are often mounted on buoys or boats and deployed in strategic locations to collect data continuously. The data collected by ADCPs is then processed to generate current profiles and maps.

Another commonly used method is the use of drifters and floats. Drifters are buoyant devices equipped with sensors that track their movement and velocity in the water. By releasing drifters at specific locations, scientists can track their movement over time, allowing them to infer the direction and speed of the currents. Floats, on the other hand, are submerged devices that use sensors to measure various parameters, including temperature and salinity, which can help in understanding the dynamics of the currents.

Data Collection and Analysis Techniques

Collecting and analyzing data is a critical step in assessing the marine current regime. In the Midia Cape-Constanta area, various techniques are employed to gather accurate and comprehensive data.

One technique involves deploying a network of monitoring buoys equipped with sensors to collect real-time data on water temperature, salinity, and currents. These buoys are strategically positioned throughout the area to provide a comprehensive understanding of the marine current regime.

Another technique is the use of numerical models. These models simulate the behavior of the currents based on various inputs, such as tidal forces, wind patterns, and topography. By comparing the model outputs with observed data, scientists can validate the accuracy of the models and gain insights into the complex dynamics of the marine currents.

Data analysis techniques, such as statistical analysis and data visualization, are then employed to interpret the collected data and identify patterns and trends in the marine current regime. These techniques help scientists and researchers make informed decisions and draw meaningful conclusions from the gathered data.

Factors Influencing Marine Currents in the Midia Cape-Constanta Area

Several factors influence the marine currents in the Midia Cape-Constanta area. Understanding these factors is crucial for accurately assessing the current regime and predicting its behavior.

One of the primary factors is tidal forces. Tides are caused by the gravitational pull of the moon and the sun, and they play a significant role in shaping the marine currents. The interaction between tidal forces and the local bathymetry determines the strength and direction of the currents.

Wind patterns also influence the marine currents. Strong winds can create surface currents, while the interaction between wind and water can generate upwelling and downwelling currents. These wind-driven currents can vary in intensity and direction, affecting the overall marine current regime.

The topography of the seabed is another crucial factor. Shallow areas, underwater ridges, and channels can alter the flow of water, creating eddies and localized currents. Understanding the bathymetry of the Midia Cape-Constanta area is essential for accurately assessing the marine current regime.

Case Study: Results of Assessing Marine Current Regime in Midia Cape-Constanta Area

To illustrate the practical application of assessing the marine current regime, let's consider a case study conducted in the Midia Cape-Constanta area. In this study, data was collected using ADCPs, drifters, and monitoring buoys, providing a comprehensive understanding of the currents in the area.

The results of the assessment revealed several key findings. Firstly, it was observed that the marine currents in this region are influenced by both tidal forces and wind patterns. The tidal currents were found to be stronger during certain periods, while the wind-driven currents varied depending on the prevailing wind direction and intensity.

The study also identified areas of high current velocities, which are of particular interest for shipping and navigation activities. By mapping these high-velocity areas, it is possible to identify potential risks and plan routes accordingly, ensuring the safety of vessels and reducing the risk of accidents or collisions.

Additionally, the assessment provided valuable insights into the movement patterns of marine organisms. By tracking the drifters and analyzing their trajectories, scientists were able to identify areas of high biological activity, which are crucial for fisheries management and conservation efforts.

Implications and Applications of the Assessment

The assessment of the marine current regime in the Midia Cape-Constanta area has several implications and applications. Firstly, the gathered information can be used to develop accurate navigational charts, helping ships and vessels navigate safely through this coastal region. By knowing the currents, captains and navigators can plan their routes more efficiently, saving time and fuel.

Secondly, the assessment provides valuable insights for fisheries management. By understanding the movement patterns of fish and other marine organisms, fisheries can optimize their fishing efforts, ensuring sustainable practices and minimizing the impact on the ecosystem. The data can also aid in the establishment of marine protected areas, protecting critical habitats and biodiversity.

Furthermore, the assessment has implications for coastal infrastructure management. By considering the impact of currents on coastal structures, engineers can design and maintain ports, jetties, and other infrastructure to withstand the forces of the currents, reducing the risk of damage and erosion.

Challenges and Limitations of Assessing Marine Current Regime

While assessing the marine current regime in the Midia Cape-Constanta area is essential, it also comes with certain challenges and limitations. One of the primary challenges is the dynamic nature of the currents. The currents can change in intensity and direction depending on various factors, making it challenging to obtain accurate and up-to-date data.

Another challenge is the complexity of the marine environment. The Midia Cape-Constanta area is influenced by multiple factors, such as tides, wind patterns, and topography, which can interact in complex ways. Understanding and modeling these interactions require sophisticated techniques and expertise.

Additionally, the cost of data collection and analysis can be a limitation. Deploying monitoring buoys, ADCPs, and other equipment can be expensive, especially for long-term monitoring projects. The availability of funding and resources can limit the scope and duration of assessments.

Future Research and Developments in Assessing Marine Current Regime

As technology and scientific understanding continue to advance, there are several exciting avenues for future research and developments in assessing the marine current regime.

One area of focus is the improvement of numerical models. By incorporating more accurate and detailed data, such as high-resolution bathymetry and weather patterns, models can provide more precise predictions of the marine current regime. The integration of artificial intelligence and machine learning algorithms can also enhance the accuracy and efficiency of these models.

Another area of research is the development of advanced sensors and monitoring techniques. Miniaturized sensors and autonomous underwater vehicles (AUVs) can provide real-time data on the currents, expanding the coverage and resolution of assessments. These advancements can help overcome the limitations of traditional monitoring methods and provide more comprehensive and accurate data.

Study 2: Interaction of Square Shaped Objects with Fluid Flow

The second study focuses on the interaction between fluid flow and square-shaped objects, an analogue for ships in open sea conditions. Employing the ANSYS CFD module, the research investigates the forces arising from pressure variation when an object interferes with fluid flow. A three-dimensional mesh around a ship model is used to analyze the interaction at varying distances and positions of the object. This study not only presents the pressure variations for each case but also offers a comprehensive view of how such interactions impact ship operation in open seas.

In the world of fluid dynamics, the interaction between objects and fluid flow is a fascinating topic. When it comes to square-shaped objects, their interaction with fluid flow brings about its own set of intriguing phenomena. From airfoils to buildings, understanding how these objects behave in fluid flows is crucial for a range of industries, including aerospace, architecture, and engineering.

In this article, we delve into the intricacies of how square-shaped objects interact with fluid flow. We will explore the principles governing the lift and drag forces acting on these objects, as well as the formation of vortices and fluid turbulence. Through a combination of theoretical analysis and experimental observations, we aim to provide a comprehensive understanding of the behavior of square-shaped objects in fluid flows.

Whether you're a researcher, engineer, or simply curious about the physics behind fluid dynamics, this article will offer valuable insights into the interaction of square-shaped objects with fluid flow. Get ready to dive into the captivating world of fluid dynamics and uncover the secrets behind the behavior of square-shaped objects.

Understanding the Interaction between Fluid Flow and Square Shaped Objects

Square-shaped objects exhibit unique characteristics when subjected to fluid flow. Unlike streamlined shapes, such as airfoils or cylinders, the interaction of square-shaped objects with fluid flow is less intuitive. The behavior of square-shaped objects is governed by the principles of lift and drag forces.

The lift force acting on a square-shaped object is responsible for its ability to generate upward forces perpendicular to the direction of fluid flow. This force is crucial for the lift and stability of various objects, including aircraft wings and wind turbine blades. The drag force, on the other hand, acts in the direction of fluid flow and creates resistance to the motion of the object.

Factors such as the angle of attack, Reynolds number, and surface roughness influence the interaction between square-shaped objects and fluid flow. The angle of attack refers to the angle at which the object is oriented with respect to the direction of fluid flow. It affects the magnitude and direction of the lift and drag forces acting on the object. The Reynolds number, a dimensionless parameter, determines the flow regime and influences the formation of vortices and turbulence around the object. Surface roughness plays a role in determining the drag force by affecting the flow separation and boundary layer characteristics.

Understanding these factors and their interplay is essential for designing square-shaped objects that perform optimally in fluid flows. It allows engineers and researchers to fine-tune the shape, orientation, and surface properties of these objects to achieve desired performance characteristics.

Experimental Studies on the Interaction of Square Shaped Objects with Fluid Flow

Experimental studies play a crucial role in unraveling the complexities of the interaction between square-shaped objects and fluid flow. Researchers use wind tunnels, water tanks, and other experimental setups to investigate the flow patterns, forces, and pressure distributions around these objects.

By instrumenting the square-shaped objects with sensors and cameras, researchers can measure the lift and drag forces, capture flow visualization, and study the formation of vortices. These experiments provide valuable data for validating theoretical models and computational fluid dynamics simulations.

One of the classic experiments conducted on square-shaped objects is the flow visualization using smoke or dye. By introducing smoke or dye into the flow, researchers can visualize the flow patterns, separation points, and vortices formed around the square-shaped object. This technique helps in understanding the complex flow behavior and identifying regions of high and low pressure.

In addition to flow visualization, researchers also employ force balance measurements to quantify the lift and drag forces acting on the square-shaped objects. These measurements provide valuable insights into the aerodynamic characteristics and allow for the optimization of object design.

Experimental studies, combined with theoretical analysis, form the foundation for understanding the behavior of square-shaped objects in fluid flows. They provide empirical data and real-world observations that complement the computational simulations and contribute to the development of accurate models.

Computational Fluid Dynamics (CFD) Simulations for Square Shaped Objects

Computational fluid dynamics (CFD) simulations have revolutionized the study of fluid flow and its interaction with square-shaped objects. CFD involves solving the governing equations of fluid flow numerically using computers. This approach allows researchers to simulate complex flow phenomena and obtain detailed information about the forces, pressure distributions, and flow characteristics.

CFD simulations require the creation of a computational domain, which encompasses the square-shaped object and the surrounding fluid. By dividing the domain into small elements or cells, the governing equations are solved iteratively to obtain the flow field. The simulations provide a wealth of information, including velocity profiles, pressure distributions, and forces acting on the object.

One of the advantages of CFD simulations is the ability to study a wide range of flow conditions and object geometries. By changing the angle of attack, Reynolds number, or surface roughness, researchers can investigate the effects of these parameters on the flow behavior. CFD simulations also allow for parametric studies, where multiple simulations are conducted to explore the design space and optimize the performance of square-shaped objects.

However, CFD simulations have their limitations. The accuracy of the simulations depends on the quality of the mesh, the choice of turbulence model, and the assumptions made in the numerical solver. Validating the simulation results with experimental data is crucial to ensure the reliability of the predictions. Nevertheless, CFD simulations have become an indispensable tool in the analysis and design of square-shaped objects in fluid flows.

Applications of the Interaction between Square Shaped Objects and Fluid Flow

The interaction between square-shaped objects and fluid flow has numerous applications in various industries. Let's explore some of the key areas where this knowledge is crucial:

1. Aerospace: The behavior of square-shaped objects, such as aircraft wings and fuselages, in fluid flows is critical for the design and performance of airplanes. Understanding the lift and drag forces, as well as the formation of vortices, plays a vital role in optimizing aircraft efficiency and stability.
2. Architecture: Square-shaped buildings and structures experience wind forces that can affect their structural integrity and occupant comfort. By understanding the interaction of these objects with fluid flow, architects can design buildings that are more aerodynamically stable and mitigate the effects of wind-induced vibrations.
3. Engineering: Square-shaped objects are commonly encountered in engineering applications, such as heat exchangers, cooling towers, and storage tanks. The interaction of these objects with fluid flow affects their efficiency, pressure drop, and overall performance. By studying this interaction, engineers can optimize the design and operation of such systems.
4. Urban Planning: The layout and arrangement of square-shaped buildings and structures in urban environments can influence the local wind patterns and create microclimates. Understanding the interaction between these objects and fluid flow helps urban planners design cities that are more energy-efficient and comfortable for the inhabitants.

These are just a few examples of the wide-ranging applications of the interaction between square-shaped objects and fluid flow. The knowledge gained from studying this interaction can lead to improved designs, enhanced performance, and more sustainable solutions in various industries.

Challenges and Limitations in Studying the Interaction of Square Shaped Objects with Fluid Flow

Studying the interaction between square-shaped objects and fluid flow presents several challenges and limitations. Let's explore some of the key obstacles researchers face in this field:

1. Complex Flow Phenomena: Square-shaped objects exhibit complex flow behavior, including the formation of vortices, flow separation, and turbulence. Understanding and predicting these phenomena accurately require advanced mathematical models and computational techniques.
2. Experimental Constraints: Conducting experiments on square-shaped objects can be challenging due to the large-scale nature of these objects. Wind tunnels and water tanks need to be designed and constructed to accommodate the size and shape of the objects, which can be costly and time-consuming.
3. Scale Effects: The behavior of square-shaped objects in fluid flow can vary depending on the size and scale of the object. Small-scale experiments may not accurately represent the flow behavior observed in real-world applications. Scaling laws and techniques are required to extrapolate the experimental data to larger scales.
4. Turbulence Modeling: Turbulence is a complex flow phenomenon that plays a significant role in the interaction of square-shaped objects with fluid flow. Accurately modeling turbulence requires the selection of appropriate turbulence models and understanding their limitations. These models introduce uncertainties that can affect the accuracy of the predictions.

Researchers in this field are continually working to overcome these challenges and develop improved methods for studying the interaction of square-shaped objects with fluid flow. By combining experimental techniques, computational simulations, and theoretical models, they aim to unravel the mysteries of fluid dynamics and enhance our understanding of this complex phenomenon.

Future Advancements and Research Directions in this Field

The study of the interaction between square-shaped objects and fluid flow is an ongoing research area with exciting prospects for future advancements. Some of the key research directions and areas of interest include:

1. Advanced Flow Visualization Techniques: Developing new visualization techniques, such as particle image velocimetry (PIV) and laser-induced fluorescence (LIF), can provide more detailed and quantitative information about the flow behavior around square-shaped objects. These techniques can help uncover additional insights into the complex flow phenomena.
2. High-Fidelity CFD Simulations: Improving the accuracy and reliability of CFD simulations is a continuous effort in the field of fluid dynamics. Advancements in numerical methods, turbulence models, and computational resources can enhance the fidelity of the simulations and provide more accurate predictions of the forces and flow characteristics.
3. Multi-Physics Simulations: The interaction of square-shaped objects with fluid flow often involves additional physical phenomena, such as heat transfer, structural dynamics, and acoustics. Integrating these multi-physics aspects into the simulations

can provide a more comprehensive understanding of the overall system behavior and enable the optimization of complex engineering designs.

4. Optimization Techniques: Developing optimization algorithms and techniques specifically tailored for square-shaped objects in fluid flow can help engineers and researchers explore the design space more efficiently. These techniques can automate the search for optimal designs based on predefined objectives and constraints.

As research in this field progresses, we can expect to see advancements in our understanding of the interaction between square-shaped objects and fluid flow. These advancements will have far-reaching implications for various industries, leading to improved designs, enhanced performance, and more sustainable solutions.

Case Studies of Real-World Examples Involving Square Shaped Objects and Fluid Flow

To illustrate the practical applications of the interaction between square-shaped objects and fluid flow, let's explore a few real-world examples:

1. High-rise Buildings: Tall square-shaped buildings are subject to strong wind forces, which can cause vibrations and discomfort for the occupants. By studying the flow behavior around these buildings, engineers can design appropriate aerodynamic features, such as tapered shapes and wind deflectors, to reduce wind-induced vibrations and improve occupant comfort.
2. Aircraft Wings: The wings of airplanes often have a square-like shape, especially near the fuselage. Understanding the flow characteristics around these wings is crucial for optimizing lift, drag, and stability. By carefully designing the wing profiles and wingtip devices, engineers can enhance the aerodynamic performance of aircraft and improve fuel efficiency.
3. Cooling Towers: Square-shaped cooling towers are commonly used in industrial applications to dissipate heat from various processes. The interaction of the cooling tower with the surrounding air affects its cooling capacity and energy consumption. By studying the flow patterns and optimizing the tower design, engineers can improve its efficiency and reduce energy consumption.

These case studies highlight the practical implications and benefits of studying the interaction between square-shaped objects and fluid flow. By gaining insights into the flow behavior and optimizing the design, engineers can achieve improved performance, enhanced efficiency, and cost savings in various applications.

Study 3: Hydrodynamics of Aquatic Systems in the Northern Romanian Sector of the Black Sea

The third study centers on the hydrodynamics of aquatic systems, particularly in the Northern Romanian sector of the Black Sea. A modeling and simulation approach reveals new perspectives on hydrological phenomena in this area. Focusing on the Danube river discharge through the Sulina branch, the study compares the CFD simulations with the CMEMS model, highlighting the advantages and limitations of each in predicting flow phenomena.

Conclusion and Future Perspectives

The synthesis of these studies offers a multifaceted understanding of marine hydrodynamics in the Black Sea region. It underscores the importance of CFD in studying complex fluid dynamics and its practical applications in navigation, hydro-technical construction, and environmental management. Future research should aim to enhance the accuracy of CFD models and extend their application to other marine environments, providing more comprehensive insights into global marine hydrodynamics.

Assessing the marine current regime in the Midia Cape-Constanta area is a complex and crucial task. By utilizing advanced technology and scientific expertise, we can gain valuable insights into the dynamics of the currents and their implications for navigation, fisheries, and coastal infrastructure management.

Through the use of tools such as ADCPs, drifters, and monitoring buoys, we can collect data on the marine currents and analyze it using various techniques. Factors such as tidal forces, wind patterns, and topography influence the marine currents in this region, and understanding these factors is essential for accurate assessment.

The results of the assessment have practical implications for shipping, fisheries, and coastal infrastructure management. By applying the knowledge gained from the assessment, we can enhance safety, optimize fishing practices, and design resilient coastal structures.

While there are challenges and limitations in assessing the marine current regime, future research and developments hold promise for improving our understanding and predictions. By continually advancing our techniques and technologies, we can better protect and utilize the marine resources in the Midia Cape-Constanta area.

The interaction between square-shaped objects and fluid flow is a captivating field of study with diverse applications in industries such as aerospace, architecture, and engineering. Understanding the lift and drag forces, the formation of vortices, and the flow behavior is crucial for optimizing the performance and efficiency of square-shaped objects.

Through experimental studies, computational fluid dynamics simulations, and theoretical analysis, researchers are continuously advancing our understanding of this complex phenomenon. Challenges such as complex flow phenomena, experimental constraints, and turbulence modeling are being addressed through innovative techniques and research efforts.

The future of this field holds promising advancements in flow visualization techniques, high-fidelity CFD simulations, multi-physics simulations, and optimization techniques. These advancements will enable engineers and researchers to design and optimize square-shaped objects in fluid flow more efficiently, leading to enhanced performance, improved designs, and sustainable solutions.

In conclusion, the interaction between square-shaped objects and fluid flow is a fascinating topic that combines physics, engineering, and mathematics. By exploring the intricacies of this interaction, we can unlock new possibilities in various industries and contribute to the advancement of science and technology.

References:

[1] Abshaev, Magomet & Abshaev, Ali & Aksenov, Andrey & Fisher, Julia & Mandous, Abdulla & Yazeedi, Omar & Wehbe, Youssef & Sîrbu, Emil & Sirbu, Andrei & Eremeico, Serghei. (2023). Abshaev et al CFD simulation of updrafts by jet 2022-Scientific Reports. Atmosphere. 30. 1 – 30.

[2] MARIN, Florin & Buruiana, Daniela & Ghisman, Viorica & Marin, Mihaela. (2023). Deep neural network modeling for CFD simulation of drone bioinspired morphing wings. INCAS BULLETIN. 15. 149-157. 10.13111/2066-8201.2023.15.4.12.

[3] Madsen, Helge. (2023). An analytical linear two-dimensional actuator disc model and comparisons with computational fluid dynamics (CFD) simulations. Wind Energy Science. 8. 1853-1872. 10.5194/wes-8-1853-2023.

[4] Li, Ruitian & Liang, Deng & Dai, Zhe & Zhang, Jian & Liu, Jie & Liu, Gang. (2023). A Data-Centric Approach for Efficient and Scalable CFD Implementation on Multi-GPUs Clusters. 10.1007/978-981-99-8211-0_10.

[5] Elgezzar, M & Rashad, A & Hassan, M & Elnady, T. (2023). CFD simulation with analytical verification of discharging of nitrogen and helium from a high-pressure gas vessel. Journal of Physics: Conference Series. 2616. 012013. 10.1088/1742-6596/2616/1/012013. Bosneagu, R., Scurtu, I. C., Popov, P., Mateescu, R., Dumitrache, L., & Mihailov, M. E. (2019). Hydraulics numerical simulation using computational fluid dynamics (CFD) method for the mouth of Sulina channel. Journal of Environmental Protection and Ecology, 20(4), 2059–2067.

[6] Bosneagu, M. R., Scurtu, I. C., Popov, P., & ... R.-D. M. (2018). Simulation on Marine currents at Midia Cape-Constanta area using computational fluid dynamics method. Thermal Science 22 (Suppl. 2), 353–360.

[7] De Angelis, Alessandro & Reinke, Nils & Ambrosini, Walter. (2023). Assessing water-wall behaviour for a light-water Small Modular Reactor with the aid of CFD analyses. Annals of Nuclear Energy. 184. 109672. 10.1016/j.anucene.2022.109672.

[8] Velasco, F.J.s. (2023). Offer: PhD position for Openfoam - CFD We have a 3years open Phd position for the research project “Development and validation of CFD combustion models for H2 and CO. Application to Severe Accident Scenarios" (OpenHyCOmb). We are looking for motivated students interested in fluiddynamics, cfd, combustion, H2, CO, high-performance computing.

[9] Song, Kelei & Ito, Kei & Ito, Daisuke & Odaira, Naoya & Saito, Yasushi & Matsushita, Kentarou & Ezure, Toshiki & Tanaka, Masaaki. (2023). CFD-BASED ANALYSIS AND EXPERIMENTAL STUDY ON GAS ENTRAINMENT PHENOMENON DUE TO FREE SURFACE VORTEX. The Proceedings of the International Conference on Nuclear Engineering (ICONE). 2023.30. 1734. 10.1299/jsmeicone.2023.30.1734.

[10] Pacheco, Jaime & Marta, André & Eça, Luis. (2023). Wind tunnel testing of a Formula Student vehicle for checking CFD simulation trends. Proceedings of the

Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering. 10.1177/09544070231203076.

[11] Scurtu, I. C., Clinci, C., & Popa, A. (2018). Water interference effect on ship due to square shaped object shielding. In IOP Conference Series: Earth and Environmental Science (Vol. 172, No. 012030). 4th International Scientific Conference SEA-CONF 2018, 17–19 May 2018, Constanta, Romania. IOP Publishing. https://doi.org/10.1088/1755-1315/172/1/012030

[12] Scurtu, I. C., & Panaitescu, V. N. (2019). Turbulent Flow Numerical Simulation for Unconventional Propulsion. Revista de Chimie, 70(10), 3508–3511.

[13] Bosneagu, R., & Scurtu, I. (2014). Weather and oceanographic influence on the maritime navigation. Constanta Maritime University Annals, Year XIV, 20.

[14] Vijaya Kumar, G.. (2023). CFD simulation of the interaction of buoyant flows with nuclear aerosols. 10.18154/RWTH-2023-10153.

[15] Boşneagu, R., Nedelcu, A. T., & Scurtu, I. C. (2018). Black Sea-the geopolitical, economic, social and military importance. Journal of Physics: Conference Series, 1122, 12006–12006.

[16] Ganiger, Manjush & Vannini, Giuseppe & Bigi, Manuele & Vidyasagar, R. & Cangioli, Filippo. (2023). CFD Analysis and Experimental Validation of a sCO2 Pocket Damper Seal. 10.1115/GT2023-102395.

[17] Romeo, B., Emil, T., & Cristian, S. I. (2021). The evolution of the temperature on the romanian Black Sea coast between 1965-2014, from the perspective of the regional and global climatic changes. IOP Conference Series: Earth and Environmental Science, 635, 12008–12008.

[18] Novac, V., Rusu, E., & Scurtu, I. C. (2019). Opportunities and risks related to offshore activities in the Western Black Sea. Journal of Environmental Protection and Ecology, 20(4), 1698–1707.

[19] Rocha, Vinicius Carvalho & Simões, Andreza & Santos, Carla & Pires, Eduardo. (2023). Hydrodynamic study of small-scale uasb reactor by computational fluid dynamics (CFD): simulation and validation. HOLOS. 5. e16400. 10.15628/holos.2023.16400.

[20] Sharma, Deepak & Ahirwar, Ritu & Pal, Shilpa. (2023). Effect of spacing on wind-induced interference on the roof of low-rise buildings with cylindrical roof using CFD simulation. Sadhana. 48. 10.1007/s12046-023-02351-5S.