Engineering Solutions to Mechanics, Marine Structures and Infrastructures

Eng. Solut. Mech. Mar. Struct. Infrastruct.
2024.1(2).2
ISSN：2960-0979
https://doi.org/10.58531/esmmsi/1/2/2

Review

The Important Role of Fluid Mechanics in the Engineering Field

Yining Zhang*, Yifan Xie, Guanyixuan Zhao, Zhengjie Liang, Jingdong Shi and Yang Yang

Ship and Maritime College, Guangdong Ocean University, Zhanjiang 524005 China

Academic Editor: Dapeng Zhang < zhangdapeng@gdou.edu.cn>
Received: 10, April, 2024; Revised: 20, April, 2024; Accepted: 23, April, 2024; Published: 24, April, 2024

Abstract: Fluid mechanics, a fundamental discipline within engineering, has gained increasing significance across various fields due to continuous advancements in engineering science and technology. Its principles and methodologies find widespread application in aerospace, energy development, environmental protection, biomedicine, and beyond, serving as a cornerstone for technological progress. However, despite the expanding applications of fluid dynamics, challenges persist in integrating theory with practice in certain domains. This paper aims to synthesize exemplary applications of fluid mechanics in diverse engineering fields, illustrating its crucial role in addressing practical challenges. Additionally, it explores current problems and limitations in fluid dynamics applications, such as numerical simulation accuracy, multiphase flow complexity, and fluid-structure interaction nonlinearity. Furthermore, recommendations are provided for future fluid mechanics development, emphasizing interdisciplinary collaboration, advanced computational methods, and the seamless integration of experimental techniques and theoretical research. Through these discussions, this paper endeavors to offer insights into the ongoing development of fluid mechanics, fostering its broader application and deeper exploration within engineering disciplines.

Keywords: Fluid mechanics; Engineering program; Experimental techniques; Theoretical research

---

Citation: Zhang Y., Xie Y., Zhao G., Liang Z., Shi J., Yang Y. The Important Role of Fluid Mechanics in the Engineering Field. Eng. Solut. Mech. Mar. Struct. Infrastruct., 2024, 1(2), doi: 10.58531/esmmsi/1/2/2

1. Introduction

As technology continues to advance and humanity's understanding of the marine world deepens, the ocean economy has emerged as a vital driver of global growth. Marine pipelines and cables play a central role in this area, serving as essential components across various sectors. As depicted in Figure 1, these pipelines and cables are integral to efficient data transmission [1], the development and utilization of marine resources [2,3], energy transportation at sea [4,5], marine fisheries [6,7], underwater robot control [8,9], sea rescue systems [10], mooring systems [11,12], marine observing systems [13], and more. Not only do they enhance operational efficiency within the marine industry, but they also provide crucial technical support for the exploration and utilization of marine resources by humanity. This particular area is garnering growing interest and attention. This particular area is garnering growing interest and attention, as shown in Figure 2.

Figure. 1. Applications of Fluid Mechanics in Engineering.

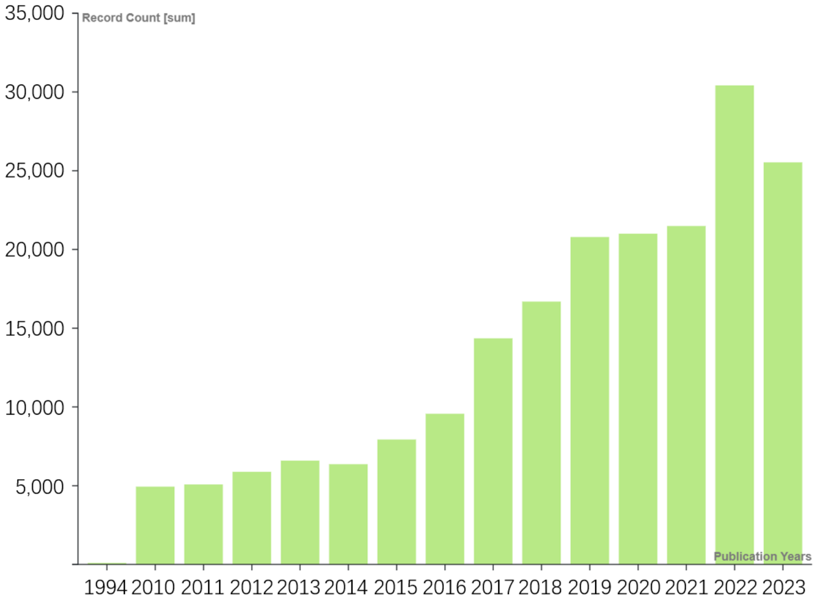

Figure. 2. The number of published results about fluid mechanics in engineering in the web of science.

This paper adopts a juxtaposed logical framework, comprehensively outlines the major categories of fluid mechanics, and briefly explains the core position of fluid mechanics in the field of engineering. By listing in detail, the practical application cases of fluid mechanics in many engineering fields, such as automobile design, aerospace, civil engineering, etc., the article profoundly demonstrates the important supportive role of fluid mechanics for engineering fields. At the same time, the article has also gained insights into the challenges and limitations faced by fluid mechanics in these fields. Based on these findings, this paper further analyzes the future development direction and trend of fluid mechanics in engineering, which provides valuable references and insights for research and practice in related fields.

2. Classification and role of fluid mechanics

There are various ways of categorizing fluid mechanics according to the research methods into computational fluid mechanics [18], theoretical fluid mechanics [19], and experimental fluid mechanics [20].

2.1 Computational Fluid Dynamics

Computational fluid mechanics, a product of advancements in computer technology, constitutes a multidisciplinary fusion of mathematics, computer science, and fluid mechanics. Primarily leveraging numerical computations on computers, computational fluid mechanics serves to analyze fluid flow, diffusion, and other properties [21]. Within the aerospace domain, computational fluid dynamics is instrumental in optimizing the aerodynamic shape of aircraft [22], while in the automotive sector, it facilitates engine combustion analysis [23]. Similarly, in the energy sector, computational fluid mechanics finds application in studying renewable energy sources [24].

The essence of computational fluid dynamics is the use of computers and numerical methods to discretize the fluid, simulate complex fluid flow, to provide, accurate and efficient analysis for the actual engineering.

2.2 Theoretical Fluid Mechanics

Theoretical fluid mechanics is an important branch in the field of fluid mechanics, he mainly through mathematical and physical methods to study the object's laws of motion, pressure characteristics, energy conversion and other information, theoretical fluid mechanics is concerned mainly with the internal mechanism of fluid flow. Theoretical fluid mechanics also has a wide range of applications within the field of engineering, theoretical fluid mechanics can be used to optimize the appearance of rockets to enhance performance, improve the efficiency of resource utilization, and optimize the flow of fluids in pipelines.

Theoretical fluid mechanics is grounded in fundamental equations such as Navier-Stokes [25] and Euler [26], which delineate the state of fluid motion and the interrelationship among its pertinent physical quantities. Through meticulous analysis of these equations, approximate solutions can be derived, offering insights into the approximate laws governing fluid motion. Such theoretical frameworks furnish invaluable references for practical applications across engineering disciplines.

2.3 Experimental Fluid Mechanics

When juxtaposed with computational and theoretical fluid mechanics, experimental fluid mechanics emerges as a more practical approach, primarily employing experimental methods to investigate the laws governing fluid motion and associated phenomena [27]. In this domain, researchers utilize an array of equipment such as flow velocity meters [27], pressure meters [28], and fluid flow visualization apparatus [29] to measure parameters including speed, pressure, and flow, among others.

Experimental fluid dynamics, theoretical fluid dynamics and computational fluid dynamics are closely related. Experimental fluid mechanics can verify the equations and principles proposed in theoretical fluid mechanics to prove the feasibility of computational fluid mechanics can be analyzed for the experimental fluid mechanics data and adjust the experimental model for the analysis results. Experimental fluid mechanics through the experimental data, for fluid mechanics provides the experimental basis, and proof of the way in the field of engineering has an irreplaceable role.

3. Fluid Mechanics in Engineering Programs

3.1 Application of fluid dynamics in the field of automotive design

With the continuous exploitation of the earth's resources by human beings, the reserves of non-renewable resources such as oil are gradually decreasing [30], a phenomenon that prompts people to seek alternative energy sources to achieve sustainable development. In this context, new energy vehicles have gradually become an important choice in the market. As a representative of new energy vehicles, the limitation of range of electric vehicles has been a key issue to be solved [31]. Especially under high-speed driving conditions, when the vehicle speed exceeds 80 km/h, the air resistance to which the vehicle is subjected will increase significantly, which leads to a sharp increase in the consumption of electric energy or fuel [32]. Therefore, for both new energy vehicles and traditional fuel vehicles, it is of vital importance to reduce wind resistance to minimize energy consumption [33]. In view of this, how to effectively reduce wind resistance has become one of the important directions of contemporary automotive research.

In 1871, British scientist Wynam achieved a significant milestone by constructing the world's first wind tunnel laboratory. This pioneering initiative laid a robust experimental foundation for the advancement of automobile design and furnished a wealth of valuable data support. Wind tunnel laboratories are capable of simulating the aerodynamic conditions experienced by a car during operation, facilitating the analysis of the resistance it encounters. Despite the considerable expense associated with their construction and operation, the indispensable role of wind tunnel laboratories in studying automotive aerodynamic characteristics cannot be overstated.

In 1984, Ahmed conducted a groundbreaking experiment on automobile wake flow [34]. This experiment effectively simulated the intricate flow dynamics around a car by employing sophisticated turbulence models such as the k-ε turbulence model [35] and the k-ωSST turbulence model [36]. These models accurately capture the turbulent kinematic properties and the energy dissipation process in detail. To streamline the simulation flow analysis while retaining key aerodynamic features of the car, Ahmed ingeniously designed a simplified car model. The specific data of the model are detailed in Table 1[37].

Table 1. Parameters related to the Ahmed model.

Characteristics	**Numerical value**
Total length（L）	1.044m
High degree（H）	0.288m
Widths（W）	0.389m
Rear Inclination	0~40°
Leg height	0.05m

The model was positioned within a flow domain measuring $8L \times 2L \times 2L$, maintaining a distance of 2L between the model and the flow port. To streamline computational efforts, the experiments incorporated the mirror symmetry technique, effectively halving the computational domain. A velocity of 40 m/s was applied at the flow port, resulting in a Reynolds number of 2.77×10^6. These meticulously designed experiments yielded a comprehensive set of key aerodynamic parameters, including pressure coefficients at the frontal, oblique, and ground surfaces.

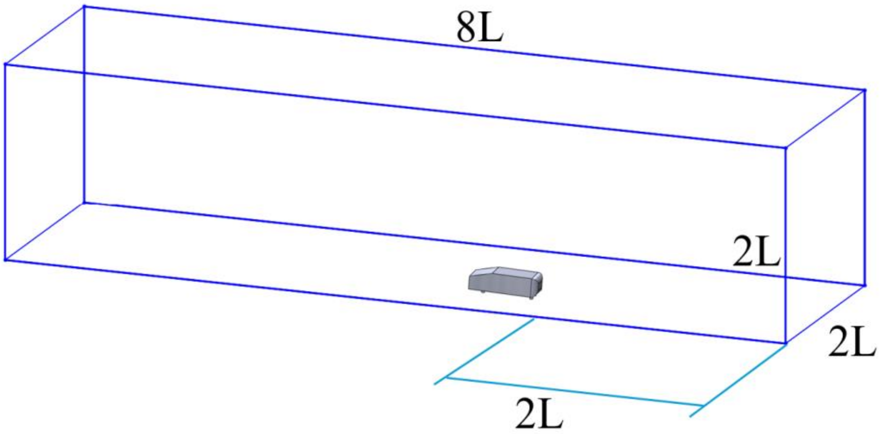

Figure. 3. Modeling of the Ahmed experiment.

Ahmed's experiments not only establish a critical theoretical and experimental foundation for automobile design but also exert a profound and enduring influence on subsequent research and optimization efforts concerning automobile aerodynamic characteristics [38]. Through these studies, automotive designers gain enhanced precision in predicting and mitigating aerodynamic drag experienced by vehicles at high speeds, thereby elevating fuel efficiency and driving performance.

In conducting this experimental study, we observed flow lines spanning the entire slant of the model, identifying two prominent return regions behind the bottom of the model. These backflow regions induce a decrease in pressure at the rear of the car, consequently amplifying the pressure differential between the front and rear ends. This differential pressure resistance constitutes a substantial portion of the aerodynamic drag components. Building upon this experimental insight, subsequent automotive design endeavors have concentrated on optimizing the shape and body structure of the vehicle's rear end to mitigate the impact of the return region and consequently reduce aerodynamic drag [39].

Specific measures encompass the adoption of a more streamlined rear end design, the incorporation of backflow control devices, and the utilization of hubcaps, among others. These design enhancements effectively diminish the proportion of the return zone in the aerodynamic drag, thereby enhancing fuel economy and driving efficiency of the vehicle [40]. Through such design optimizations, we achieve significant enhancements in automobile performance at high speeds while concurrently reducing energy consumption, a pivotal step in promoting the sustainable development of the automotive industry.

3.2 Applications of Fluid Mechanics in Aerospace

The aerospace sector stands as a pivotal domain of scientific and technological advancement globally [41], playing a critical role in both transportation and national defense security. Within this realm, the lift-to-drag ratio and flight stability of a vehicle serve as crucial performance indicators [42]. Given the prohibitively high costs associated with manufacturing full-scale experimental models for field testing aerospace vehicles, practicality necessitates alternative approaches.

However, with the continual evolution of computational fluid dynamics (CFD) technology, CFD software platforms such as ANSYS Fluent [43], OpenFOAM [44], and Phoenics [45] have emerged as indispensable analytical tools in aerospace engineering. These advanced computational tools enable engineers to simulate and analyze complex aerodynamic phenomena with unprecedented accuracy and efficiency, facilitating the optimization of aerospace vehicle designs and enhancing their performance characteristics.

These advanced CFD software packages excel in accurately simulating the intricate fluid dynamics phenomena encountered by aerospace vehicles during flight, enabling comprehensive analysis of the flow field [46] and evaluation of aerodynamic characteristics [47]. Through CFD simulations, engineers gain the capability to predict vehicle performance during the design phase [48], identify potential issues, and execute necessary optimizations, thereby significantly enhancing design efficiency and curtailing development costs. Moreover, the application of CFD techniques catalyzes innovation by facilitating the exploration of novel aerodynamic designs and materials, ultimately striving toward higher flight efficiency and superior flight performance.

Hence, the utilization of CFD technology in the aerospace domain not only furnishes robust data support for aircraft design and optimization but also serves as a potent analysis platform for researchers and engineers in this field. It enables effective aerodynamic characterization studies and performance evaluation of aircraft without compromising accuracy. With the ongoing enhancement of computational power and the continuous refinement of algorithms, the application of CFD technology in aerospace holds a promising outlook [49], paving the way for new possibilities in the future development of aerospace technology.

3.3 Application of hydrodynamics in the field of ships

With the ocean encompassing 71% of the Earth's surface, the vitality of the ship industry holds immense significance, particularly for China, endowed with an extensive coastline. This industry is pivotal for both national defense security and maritime economic development [50]. Fluid mechanics, serving as a cornerstone in ship design and engineering [51], assumes a central role in analyzing and optimizing various aspects of ship performance. This includes the reduction of ship resistance [52], enhancement of propulsion efficiency [53], improvement of maneuverability [54], and mitigation of noise and vibration.

Through a detailed analysis of the interaction between propellers and hydrodynamics, propeller design can be optimized to achieve higher propulsive efficiency and lower noise levels. Furthermore, hydrodynamic principles are instrumental in studying ship maneuverability, enabling designers to anticipate and enhance the response characteristics of ships in complex waters. Moreover, hydrodynamics plays a crucial role in ship noise and vibration control [55]. By examining the noise and vibration resulting from fluid flow, effective measures can be implemented, such as optimizing hull structures and utilizing damping materials, to diminish noise and vibration during ship operation, thereby enhancing the comfort of occupants and cargo.

Zhirong Zhang and his team conducted a successful simulation study on the hydrodynamic performance of a ship, employing the three-dimensional Reynolds-averaged Navier-Stokes (RANS) equations as the governing equations [56]. These equations were solved using numerical methods, resulting in comprehensive insights into the ship's hydrodynamic behavior.

In the study by Zhang Zhirong et al, the finite volume method was utilized, employing a linear reconstruction scheme for numerical solution. This method offers the advantage of flexibility in handling computational cells with arbitrary polyhedral topology. To enhance the accuracy of the numerical solution, convective terms were discretized using a second-order windward scheme, while the velocity-pressure coupling problem was addressed using the SIMPLE algorithm [57]. By employing the pointwise Gauss-Seidel iterative algorithm [58], efficient solution of the discretized algebraic equations was achieved. Furthermore, the introduction of the algebraic multigrid (AMG) method [58] served to expedite the convergence of the solution process.

In solving the free-surface boundary problem, Zhirong Zhang's research team used the "volume of fluid" (VOF) method [59]. This method skillfully simulates the morphology and dynamics of a free surface by tracking the volume fractions of different fluids in the computational domain. Using the VOF method, the researchers performed a comprehensive numerical simulation of the ship model, which enabled a detailed assessment of the drag and wave generation of the hull under different Froude number conditions.

Through meticulous comparison and analysis of numerical simulation results with experimental data across various parameters such as wave contours, profiles, and drag coefficients, the research team discerned a remarkably high degree of consistency between the two. This outcome not only corroborates the accuracy of the numerical simulation method but also underscores the immense potential of Computational Fluid Dynamics (CFD) technology in modeling the hydrodynamic performance of ships. The findings of this study establish a robust foundation for further exploration in the realm of ship design and hydrodynamics, heralding new opportunities for future engineering practices and scientific research endeavors.

3.4 Application of Fluid Mechanics in Civil Engineering

Civil engineering, intimately intertwined with human life [60], encompasses critical domains such as urban construction, water conservancy engineering, and transportation engineering. In these realms, the principles and applications of fluid mechanics assume paramount importance [61], furnishing a sturdy theoretical framework for engineering design, construction, and maintenance.

Within urban drainage systems, fluid dynamics plays a pivotal role in optimizing pipeline network design to facilitate efficient water conveyance and wastewater treatment [62]. Moreover, it underpins the development of strategies for rainwater harvesting and flood prevention and control. By meticulously calculating fluid dynamics properties, engineers can devise drainage facilities that are both cost-effective and efficient.

In hydraulic engineering, particularly in hydroelectric power generation [63], fluid dynamics is indispensable for analyzing and enhancing energy conversion efficiency. Similarly, in transportation engineering, the construction of bridges relies heavily on fluid dynamics. Hydrodynamic analysis of bridge structures aids in evaluating and mitigating the impact of wind on bridge integrity, thereby ensuring structural safety under diverse wind conditions.

When heavy trucks traverse bridge structures, they may encounter significant variations in aerodynamic forces, presenting a grave safety hazard for drivers and passengers due to the potential for vehicle instability or rollover [64]. To comprehensively understand this phenomenon and assess its impact on traffic safety, researchers have employed computational fluid dynamics (CFD) techniques to simulate the aerodynamic behavior of heavy trucks downstream of bridge towers [65]. The dynamic mesh technique was effectively utilized to simulate the relative motion between the vehicle and the bridge tower, thereby accurately capturing the intricate interaction and aerodynamic effects between the two entities.

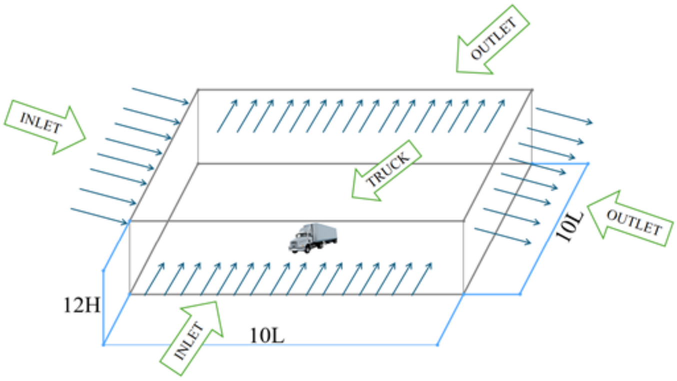

Figure. 4. Pneumatic simulation of trucks.

In order to verify the accuracy of the simulation, a comprehensive comparison and analysis of the CFD simulation results and wind tunnel experimental data was conducted by Zhang et al. The experimental findings unveil a notable phenomenon: when a heavy truck traverses a bridge tower, it undergoes substantial aerodynamic alterations. Particularly noteworthy is the sharp increase in lateral force experienced by the vehicle upon exiting the wake region of the bridge tower. This pronounced increase significantly heightens the risk of traffic accidents [66].

Based on these findings, Zhang et al. proposed a feasible solution: incorporating lateral shielding into the design of existing infrastructures. Such a design modification can effectively mitigate the impact of lateral winds, consequently reducing the risk of accidents and bolstering the safety of heavy trucks navigating through intricate bridge structures. This strategy not only enhances traffic safety but also offers a novel approach to bridge and roadway design, furnishing a scientific foundation for future infrastructure planning and safety enhancements.

3.5 Fluid Mechanics in Film and Television Engineering

The utilization of fluid dynamics in film and television production has emerged as a pivotal technology for enhancing visual effects and augmenting audience immersion [67]. In animation and television production, precise modeling of dynamic liquid and gas flows facilitates the creation of highly realistic visual effects, thereby significantly enriching the expressive capabilities of visual narratives. Similarly, in the realm of game development, the application of fluid dynamics holds equal importance [68]. Employing fluid mechanics to simulate phenomena such as smoke propagation, water dynamics, and flame combustion [69] not only elevates the visual appeal of the game but also enhances the player's sense of immersion.

The application of fluid dynamics in the film, television, and gaming industries is experiencing a surge in sophistication and efficiency, owing to the escalation in computing power and the relentless advancement of simulation technology. As research and development endeavors in fluid dynamics within the film and television industry persist in driving the innovation of visual effects, audiences can anticipate an enhanced visual experience and a more immersive interactive journey.

3.6 Applications of Fluid Mechanics in Medical Engineering

The applications of fluid dynamics in medicine are burgeoning, playing pivotal roles in advancing diagnostic imaging techniques [70] and reshaping drug delivery systems [71]. The integration of computational fluid dynamics (CFD) into magnetic resonance imaging (MRI) and computed tomography (CT) has notably enhanced imaging quality and accuracy [72]. By meticulously modeling the behavior of fluid dynamics within biological tissues, CFD enables medical professionals to observe and comprehend physiological and pathological processes with greater clarity [73].

The utilization of fluid dynamics principles has demonstrated significant potential in the design of nano drug delivery robots [74]. These micro-robots possess the capability to deliver drugs precisely to the site of the lesion, thereby enhancing drug efficacy and minimizing side effects on healthy tissues [75]. Moreover, fluid dynamics simulation technology has opened up new avenues in surgical procedures. By simulating changes in biofluid dynamics during surgery, physicians can forecast surgical outcomes, evaluate potential risks, and formulate more precise and safer surgical protocols [76].

In-depth analysis of blood flow through computational fluid dynamics (CFD) simulation has emerged as a vital tool for addressing complex biofluid dynamics challenges. Notably, CFD simulation offers an efficient alternative approach to investigating blood flow characteristics and refining surgical protocols, particularly in scenarios where direct experimentation is impractical due to small target areas or inherent complexities [77]. By precisely simulating the flow dynamics of blood within the vasculature, we can discern subtle alterations in hemodynamics and forecast potential blood flow patterns. Such insights are crucial for devising precise surgical strategies.

Following the validation of the optimal treatment plan through CFD simulation, the integration of micro-robotics technology further elevates the precision and safety of treatment [78]. These sophisticated robotic systems exhibit remarkable capabilities in precise navigation to the target treatment area and executing delicate maneuvers under the surgeon's control, thereby substantially enhancing the success rate of surgeries. Moreover, this synergy of technologies contributes to minimizing surgical trauma and accelerating patient recovery time, culminating in a safer and more efficacious healthcare experience for patients.

In conclusion, the fusion of CFD simulation and micro-robotics not only offers an innovative solution to the challenges inherent in the medical field, but also heralds new horizons for precision medicine and minimally invasive surgery in the future. Through this interdisciplinary amalgamation of technologies, we gain enhanced insights into and control over the intricacies of blood flow dynamics, thereby enabling the provision of more precise and personalized treatment plans for patients.

3.7 Applications of Fluid Mechanics in Marine Engineering

The ocean harbors abundant resources [79], underscoring the significance of ocean engineering for their comprehensive utilization. Constructing marine platforms enables the exploitation of oceanic resources, with targeted construction enhancements based on fluid state simulations near these platforms [80] to bolster their structural integrity and longevity. Moreover, in cases where oceanic resource development leads to pollution, hydrodynamics can be leveraged to simulate ocean current flow states [81], facilitating the prediction of pollutant diffusion and migration. Subsequently, appropriate measures can be implemented to mitigate pollutant effects, guided by these simulations.

Crude oil spills, although rare in marine oil development activities, wield significant potential to disrupt marine ecosystems [82]. Notable incidents such as those in the Gulf of Mexico and the Penglai oil field in the Bohai Sea have underscored the severe environmental pollution that can result from such occurrences. Hence, timely prediction and response to crude oil spills are imperative [83], particularly in the initial stages, where precise pollution propagation forecasting is pivotal for containment and environmental damage mitigation.

In this regard, computational fluid dynamics (CFD) simulation coupled with the volume fraction (VOF) method emerges as an effective tool for predicting the trajectory and dispersion pattern of spilled crude oil. Researchers have conducted a series of CFD simulations utilizing FLUENT [84] software, which incorporates various influencing factors including crude oil density, leakage rate, and water velocity. This enables the dynamic simulation of crude oil leakage from submarine pipelines into seawater [85].

The simulation outcomes illustrate that post-crude oil spill, the oil experiences a complex interplay of gravitational, inertial, buoyancy, and shear forces during its ascent, leading to dispersion of oil droplets in seawater. Moreover, the time taken for oil droplets to reach the sea surface correlates closely with crude oil density and spill rate, while the horizontal propagation distance of oil droplets positively correlates with current velocity [86].

Based on these insights, researchers derived dimensionless formulas to depict the time required for an oil droplet to reach the sea surface and modeled the prediction of the maximum horizontal propagation distance of an oil droplet at a given time. The application of these models facilitates the anticipation of potential oil droplet distribution on the sea surface following a crude oil spill, thereby guiding the proactive deployment of oil containment barriers and effectively curbing further pollution spread.

The oil droplet uplift simulation experiment highlights the great potential of CFD simulation combined with the VOF method in predicting and treating submarine crude oil spills, which provides a strong technical support for marine environmental protection. Through these advanced simulation techniques, we gain enhanced insights into the behavior of spilled crude oil, enabling the provision of scientifically informed and timely countermeasures. Consequently, we can mitigate the damage to the marine environment more effectively, underscoring the pivotal role of such methodologies in safeguarding marine ecosystems.

3.8 Applications of fluid mechanics in the field of acoustics

Sound propagation relies on the presence of a medium, with fluids—comprising gases and liquids—serving as critical conduits in this process [87]. In practical applications, integrating the principles of fluid mechanics and acoustics proves instrumental in addressing a spectrum of intricate challenges associated with sound propagation. For instance, by accurately modeling airflow dynamics in urban settings and fluid interactions among buildings, we can forecast and assess noise propagation patterns in urban environments [88]. Furthermore, leveraging fluid dynamics principles allows for a comprehensive understanding and optimization of indoor acoustic properties, facilitating the design of indoor spaces with enhanced acoustic performance. This approach not only enhances our comprehension of sound behavior in complex fluid environments but also holds significant practical implications for urban planning, architectural design, and noise control.

Amid increasing attention to the health and welfare of marine ecosystems, the influence of underwater noise has emerged as a significant consideration in ship design [89]. To tackle this challenge, researchers have adopted an integrated numerical simulation approach, which combines the unsteady Reynolds-averaged Navier-Stokes (URANS) equations for hydrodynamic prediction with the Ffowcs-Williams Hawkings (FWH) equations [90] to model noise propagation, as shown in equation (1).

$$\nabla^2 p' = \frac{\partial}{\partial t}\left[\rho_0 v_n \delta(f)\right] - \frac{\partial}{\partial x_i}[1_i \delta(f)] + \frac{\bar{\partial}^2}{\partial x_i \partial y_j}[T_{ij} H(f)] \tag{1}$$

The method is able to record in detail the noise levels above and below water at different speed conditions, providing important acoustic data for ship design [91].

To ensure the precision of the simulation experiments, the research team meticulously accounted for a multitude of modeling variables to accurately mirror real-world scenarios. Introducing a permeable source surface in the flow field facilitated the simulation of noise propagation characteristics in the fluid. Furthermore, in representing propellers, the study conducted a comparative analysis of various techniques including rotating grids, moving reference frames, and static methods to assess their respective merits and drawbacks in terms of accuracy, cost, and computational demands.

To authentically replicate the dynamic behavior of free surfaces, studies have employed volumetric fluid (VOF) models and Eulerian multiphase models [92]. These models excel in precisely capturing the intricate phenomena of free-surface-fluid interaction, thereby facilitating accurate prediction of noise propagation.

Through these advanced numerical simulation techniques, researchers were able to develop an accurate noise prediction model. This model can not only provide guidance for current ship design, but also provide a scientific basis for the development of future ship noise control and mitigation strategies [93], thus protecting marine life from noise interference while promoting ship design in a more environmentally friendly and sustainable direction.

4. Challenges and Limitations of Fluid Mechanics in Engineering Applications

Fluid mechanics assumes a pivotal role across various engineering domains, particularly within the aerospace industry, where it remains indispensable for vehicle design and performance optimization [94]. Nevertheless, despite remarkable strides in computational technology that have notably enhanced our capacity to simulate intricate fluid dynamics, the real flight conditions entail an environment rife with uncertainties and variables. Present computational resources continue to grapple with the dual challenges of accuracy and efficiency when confronted with complex flow problems at the industrial scale [95]. Although turbulence plays an important role in the field of fluid mechanics, there is still a lack of a complete theoretical framework to depict the complexity of turbulence, and the existing turbulence models can only perform well in simple cases and have the disadvantage of lack of accuracy in complex cases.

Moreover, the high cost associated with experimental hydrodynamics research [96] and the intricate nature of experimental conditions pose constraints on enhancing accuracy. While the acquisition of experimental data is pivotal for validating theories and models, replicating the full complexity of actual flight conditions remains a formidable challenge in fluid dynamics research.

Addressing multiscale problems presents a technical challenge in the field of fluid mechanics [97]. Fluid phenomena encompass a range of scales, spanning from microscopic molecular dynamics to macroscopic hydrodynamic behaviors. Establishing effective coupling and conversion mechanisms between these diverse scales remains a focal point of current research efforts.

Applications of fluid mechanics frequently extend beyond isolated domains, often integrating with other disciplines such as structural engineering, thermal sciences, and various related fields [98-100]. This interdisciplinary integration not only amplifies the complexity of problem-solving endeavors but also imposes elevated knowledge requirements on engineers and researchers. Consequently, the effective application of fluid mechanics principles in these cross-cutting domains has emerged as pivotal for propelling technological progress and fostering innovation.

5. Trends in fluid mechanics

With the rapid advancement of computing technology, the computational power of fluid dynamics is seeing an unprecedented increase. This means that we can more accurately simulate the complex and variable fluid flow situation in the real world, and the fluid behavior in the virtual world is infinitely close to the fluid behavior in the real world. This progress not only brings a broader space for the application of simulation software, but also significantly saves the cost of actual experiments and improves the efficiency of scientific research and engineering development. In current research, multi-scale modeling has become one of the key directions in the development of fluid mechanics. It requires researchers to develop simulation methods that can effectively connect the microscale (e.g., molecular dynamics) with the macroscale (e.g., continuous medium behavior). This cross-scale simulation method not only helps us to predict the flow behavior of fluids more accurately, but also reveals the intrinsic mechanisms of complex phenomena in fluids, providing a more comprehensive perspective for understanding the nature of fluid behavior.

In addition, as a basic discipline, fluid mechanics is gradually integrating with other disciplines, such as structural mechanics, thermodynamics, biomedicine and so on. This cross-fertilization not only broadens the research field of fluid mechanics, but also provides new ideas and methods to solve more complex engineering and scientific problems [101]. For example, in the field of biomedicine, the application of fluid mechanics can help us better understand the flow of blood in blood vessels and provide a more scientific basis for the treatment and prevention of diseases.

Looking ahead, this interdisciplinary collaboration will promote the deepening of the application of fluid mechanics in multiple fields and the development of innovative technologies. We have reason to believe that with the continuous progress of computational technology and the deepening of cross-disciplinary integration, fluid mechanics will play an even more important role in the future, and provide more effective tools and means for solving the complex problems faced by mankind.

Acknowledgments: I extend my heartfelt gratitude to my supervisor, Dr. Dapeng Zhang, whose unwavering support and expert guidance have been instrumental in shaping my journey through scientific research and in meticulously steering me through the process of writing my thesis. His invaluable mentorship has not only enhanced my academic prowess but has also instilled in me a deeper passion for inquiry and discovery.

I am deeply indebted to my parents for their boundless love, unwavering encouragement, and unwavering belief in my abilities. Their steadfast support and sacrifices have been the cornerstone of my academic pursuits, providing me with the strength and determination to overcome challenges and pursue excellence.

To my cherished friends, I express my sincere appreciation for their companionship, encouragement, and unwavering belief in my endeavors. Their camaraderie and unwavering support have been a source of solace and inspiration, enriching my academic journey with laughter, shared experiences, and cherished memories.

In conclusion, I am profoundly grateful to Dr. Zhang, my parents, and my friends for their invaluable contributions to my academic and personal growth. Their unwavering support, guidance, and encouragement have been the driving force behind my achievements, and I am forever grateful for their presence in my life.

Funding: This research was funded by Program for Scientific Research Start-up Funds of Guangdong Ocean University, grant number 060302112008, Zhanjiang Marine Youth Talent Project- Comparative Study and Optimization of Horizontal Lifting of Subsea Pipeline, grant number 2021E5011 and the National Natural Science Foundation of China, grant number 62272109.

Conflict of interest: The authors of the article have consulted with each other and have no conflict of interest.

References:

1. Zhang, Yi, Dapeng Zhang, and Haoyu Jiang. "Review of Challenges and Opportunities in Turbulence Modeling: A Comparative Analysis of Data-Driven Machine Learning Approaches." Journal of Marine Science and Engineering 11.7 (2023): 1440.
2. Cao, Leo NY, Zijie Xu, and Zhong Lin Wang. "Application of triboelectric

nanogenerator in fluid dynamics sensing: past and future." Nanomaterials 12.19 (2022): 3261.

3. Cermak, Jack E. "Applications of fluid mechanics to wind engineering—a Freeman Scholar lecture." (1975): 9-38.
4. Wootton, David M., and David N. Ku. "Fluid mechanics of vascular systems, diseases, and thrombosis." Annual review of biomedical engineering 1.1 (1999): 299-329.
5. Kay, John Menzies, and Ronald Midgley Nedderman. An Introduction to Fluid Mechanics and Heat Transfer: with applications in chemical and mechanical process engineering. CUP Archive, 1974.
6. Chen, Xi, Baoxing Xu, and Ling Liu. "Nanoscale fluid mechanics and energy conversion." Applied Mechanics Reviews 66.5 (2014): 050803.
7. Whitaker, Stephen. "The development of fluid mechanics in chemical engineering." One Hundred Years of Chemical Engineering: From Lewis M. Norton (MIT 1888) to Present. Dordrecht: Springer Netherlands, 1989. 47-109.
8. Peacock, Thomas, and Raphael Ouillon. "The fluid mechanics of deep-sea mining." Annual Review of Fluid Mechanics 55 (2023): 403-430.
9. Shuja, S. B., and R. M. Raja. "PARICALCITOL IMPROVES ERYTHROPOIETIN RESPONSIVENESS." ASAIO Journal 49.2 (2003): 194.
10. Arndt, Roger EA. "Cavitation in fluid machinery and hydraulic structures." Annual Review of Fluid Mechanics 13.1 (1981): 273-326.
11. Aliabadi, Shahrouz K., and Tayfun E. Tezduyar. "Parallel fluid dynamics computations in aerospace applications." International Journal for Numerical Methods in Fluids 21.10 (1995): 783-805.
12. Godunov, Sergei K., and I. Bohachevsky. "Finite difference method for numerical computation of discontinuous solutions of the equations of fluid dynamics." Matematičeskij sbornik 47.3 (1959): 271-306.
13. Shen, S-F. "Finite-element methods in fluid mechanics." Annual Review of Fluid Mechanics 9 (1977): 421-445.
14. Onate, E., et al. "A stabilized finite point method for analysis of fluid mechanics problems." Computer methods in applied mechanics and engineering 139.1-4 (1996): 315-346.
15. Widnall, Sheila E. "The structure and dynamics of vortex filaments." Annual Review of Fluid Mechanics 7.1 (1975): 141-165.
16. Makinde, Oluwole Daniel, Waqar Ahmed Khan, and Tirivanhu Chinyoka. "New developments in fluid mechanics and its engineering applications." Mathematical Problems in Engineering 2013 (2013).
17. Bhatti, Muhammad Mubashir, et al. "Recent trends in computational fluid dynamics." Frontiers in Physics 8 (2020): 593111.
18. Bhatti, Muhammad Mubashir, et al. "Recent trends in computational fluid dynamics." Frontiers in Physics 8 (2020): 593111.
19. Lighthill, James. "An informal introduction to theoretical fluid mechanics." (1986).
20. Wang, Ruey-Wen, Laurent David, and Michel Larinier. "Contribution of experimental fluid mechanics to the design of vertical slot fish passes." Knowledge and Management

of Aquatic Ecosystems 396 (2010): 02.

21. Ashgriz, Nasser, and Javad Mostaghimi. "An introduction to computational fluid dynamics." Fluid flow handbook 1 (2002): 1-49.
22. Agarwal, Ramesh. "Computational fluid dynamics of whole-body aircraft." Annual review of fluid mechanics 31.1 (1999): 125-169.
23. D'Errico, Gianluca, et al. "Computational fluid dynamics simulation of diesel engines with sophisticated injection strategies for in-cylinder pollutant controls." Energy & fuels 26.7 (2012): 4212-4223.
24. Teimouri, Zahra, et al. "Application of computational fluid dynamics for modeling of Fischer-Tropsch synthesis as a sustainable energy resource in different reactor configurations: A review." Renewable and Sustainable Energy Reviews 160 (2022): 112287.
25. Glowinski, Roland, and Olivier Pironneau. "Finite element methods for Navier-Stokes equations." Annual review of fluid mechanics 24.1 (1992): 167-204.
26. Trac, Hy, and Ue-Li Pen. "A primer on eulerian computational fluid dynamics for astrophysics." Publications of the Astronomical Society of the Pacific 115.805 (2003): 303.
27. Adrian, Ronald J. "Particle-imaging techniques for experimental fluid mechanics." Annual review of fluid mechanics 23.1 (1991): 261-304.
28. Taylor, Charles A., and Mary T. Draney. "Experimental and computational methods in cardiovascular fluid mechanics." Annu. Rev. Fluid Mech. 36 (2004): 197-231.
29. Perissinotto, Rodolfo Marcilli, et al. "Flow visualization in centrifugal pumps: A review of methods and experimental studies." *Journal of Petroleum Science and Engineering* 203 (2021): 108582.
30. Deng, Xiangzheng, et al. "Depletable and Non-renewable Energy Resources: Coal and Oil Resources." Environmental and Natural Resources Economics. Singapore: Springer Nature Singapore, 2024. 263-301.
31. Yuan, Quan, et al. "Investigation on range anxiety and safety buffer of battery electric vehicle drivers." Journal of Advanced Transportation 2018 (2018).
32. Chang, David J., and Edward K. Morlok. "Vehicle speed profiles to minimize work and fuel consumption." Journal of transportation engineering 131.3 (2005): 173-182.
33. Chang, David J., and Edward K. Morlok. "Vehicle speed profiles to minimize work and fuel consumption." Journal of transportation engineering 131.3 (2005): 173-182.
34. Igali, Dastan, et al. "Comparative analysis of turbulence models for automotive aerodynamic simulation and design." International Journal of Automotive Technology 20 (2019): 1145-1152.
35. Saeed, Muhammad, et al. "An assessment of k-ε turbulence models for gas distribution analysis." *Nuclear Science and Techniques* 28 (2017): 1-8.
36. Hu, Peng, et al. "Numerical simulations of the mean wind speeds and turbulence intensities over simplified gorges using the SST k-ω turbulence model." Engineering Applications of Computational Fluid Mechanics 10.1 (2016): 359-372.
37. Dumas, Laurent. "CFD-based optimization for automotive aerodynamics." Optimization and computational fluid dynamics. Berlin, Heidelberg: Springer Berlin Heidelberg, 2008.

191-215.

38. Le Good, Geoffrey M., and Kevin P. Garry. "On the use of reference models in automotive aerodynamics." (2004).
39. Hassan, SM Rakibul, et al. "Numerical study on aerodynamic drag reduction of racing cars." Procedia Engineering 90 (2014): 308-313.
40. Janssen, L. J., and H-J. Emmelmann. "Aerodynamic Improvements-A Great Potential for Better Fuel Economy." SAE Transactions (1978): 1261-1269.
41. Roman-Gonzalez, Avid. "The UNSAAC activities in aerospace development." 65th International Astronautical Congress–IAC 2014. 2014.
42. Qu, Qiulin, et al. "Numerical study of aerodynamics of a wing-in-ground-effect craft." Journal of Aircraft 51.3 (2014): 913-924.
43. Pashchenko, D. I. "ANSYS fluent CFD modeling of solar air-heater thermoaerodynamics." Applied Solar Energy 54 (2018): 32-39.
44. Higuera, Pablo, Javier L. Lara, and Inigo J. Losada. "Simulating coastal engineering processes with OpenFOAM®." Coastal Engineering 71 (2013): 119-134.
45. Rosten, H. I., D. B. Spalding, and D. G. Tatchell. "PHOENICS: a general-purpose program for fluid-flow, heat-transfer and chemical-reaction processes." Engineering Software III: Proceedings of the 3rd International Conference, Imperial College, London, England. April 1983. Berlin, Heidelberg: Springer Berlin Heidelberg, 1983.
46. Tominaga, Yoshihide, and Ted Stathopoulos. "Turbulent Schmidt numbers for CFD analysis with various types of flowfield." Atmospheric Environment 41.37 (2007): 8091-8099.
47. Dantsker, Or, and Moiz Vahora. "Comparison of aerodynamic characterization methods for design of unmanned aerial vehicles." 2018 AIAA Aerospace Sciences Meeting. 2018.
48. Morgado, J., et al. "XFOIL vs CFD performance predictions for high lift low Reynolds number airfoils." Aerospace Science and Technology 52 (2016): 207-214.
49. va, Bram. "CFD education-Past, present, future." 37th Aerospace Sciences Meeting and Exhibit. 2000.
50. Para-González, Lorena, and Carlos Mascaraque-Ramírez. "The importance of official certifications in globalized companies' performance: An empirical approach to the shipbuilding industry." Corporate Social Responsibility and Environmental Management 26.2 (2019): 408-415.
51. Chirosca, Ana-Maria, and Liliana Rusu. "The Use of CFD Methods in the Shipbuilding Industry and their Benefits." Acta Universitatis Danubius. Œconomica 17.6 (2021).
52. Campana, Emilio F., et al. "Shape optimization in ship hydrodynamics using computational fluid dynamics." Computer methods in applied mechanics and engineering 196.1-3 (2006): 634-651.
53. Jiang, Jingwei, et al. "A ship propeller design methodology of multi-objective optimization considering fluid–structure interaction." Engineering Applications of Computational Fluid Mechanics 12.1 (2018): 28-40.
54. Li, Ye, et al. "Improving ship maneuverability standards as a means for increasing ship controllability and safety." Proceedings of OCEANS 2005 MTS/IEEE. IEEE, 2005.
55. Viitanen, Ville, Antti Hynninen, and Tuomas Sipilä. "Computational fluid dynamics and

hydroacoustics analyses of underwater radiated noise of an ice breaker ship." Ocean Engineering 279 (2023): 114264.

56. Zhang, Zhi-rong, et al. "Application of CFD in ship engineering design practice and ship hydrodynamics." Journal of Hydrodynamics, Ser. B 18.3 (2006): 315-322.
57. Latimer, B. R., and A. Pollard. "Comparison of pressure-velocity coupling solution algorithms." Numerical Heat Transfer 8.6 (1985): 635-652.
58. Edalatpour, Vahid, Davod Hezari, and Davod Khojasteh Salkuyeh. "A generalization of the Gauss–Seidel iteration method for solving absolute value equations." Applied Mathematics and Computation 293 (2017): 156-167.
59. Garoosi, Faroogh, and Kamel Hooman. "Numerical simulation of multiphase flows using an enhanced Volume-of-Fluid (VOF) method." International Journal of Mechanical Sciences 215 (2022): 106956.
60. Russell, Jeffrey S. "Shaping the future of the civil engineering profession." Journal of construction engineering and management 139.6 (2013): 654-664.
61. Yetilmezsoy, Kaan. "IMECE—Implementation of mathematical, experimental, and computer-based education: A special application of fluid mechanics for civil and environmental engineering students." Computer Applications in Engineering Education 25.5 (2017): 833-860.
62. Dewals, Benjamin J., et al. "Semi-explicit modelling of watersheds with urban drainage systems." Engineering Applications of Computational Fluid Mechanics 6.1 (2012): 46-57.
63. Shin, Hyun Ho. "A methodology of study of three dimensional stratified turbulent fluid flow for hydroelectric power plant reservoir simulation." (2009).
64. Salati, Luigi, et al. "Aerodynamic study on a heavy truck passing by a bridge pylon under crosswinds using CFD." Journal of Bridge Engineering 23.9 (2018): 04018065.
65. Bayraktar, Ilhan, and Tuba Bayraktar. Guidelines for cfd simulations of ground vehicle aerodynamics. No. 2006-01-3544. SAE Technical Paper, 2006.
66. Han, Yan, et al. "Effects of sudden change in wind loads on running performance of trains on a highway-railway one-story bridge in crosswinds." Journal of Central South University 29.8 (2022): 2621-2638.
67. Nanduri, Jagannath R., Francisco A. Pino-Romainville, and Ismail Celik. "CFD mesh generation for biological flows: Geometry reconstruction using diagnostic images." Computers & Fluids 38.5 (2009): 1026-1032.
68. Deng, Yujuan. "Fluid Equation-Based and Data-Driven Simulation of Special Effects Animation." Advances in Mathematical Physics 2021 (2021): 1-11.
69. Foster, Nicholas D. Modeling fluid phenomena for computer graphics and special effects. University of Pennsylvania, 1997.
70. Nanduri, Jagannath R., Francisco A. Pino-Romainville, and Ismail Celik. "CFD mesh generation for biological flows: Geometry reconstruction using diagnostic images." Computers & Fluids 38.5 (2009): 1026-1032.
71. Lindner, Wolfram D., and Bernhard C. Lippold. "Drug release from hydrocolloid embeddings with high or low susceptibility to hydrodynamic stress." Pharmaceutical research 12 (1995): 1781-1785.

72. Mastrodicasa, Domenico, et al. "Artificial intelligence machine learning-based coronary CT fractional flow reserve (CT-FFRML): Impact of iterative and filtered back projection reconstruction techniques." Journal of Cardiovascular Computed Tomography 13.6 (2019): 331-335.
73. Taylor, Charles A., and David A. Steinman. "Image-based modeling of blood flow and vessel wall dynamics: applications, methods and future directions: Sixth International Bio-Fluid Mechanics Symposium and Workshop, March 28–30, 2008 Pasadena, California." Annals of biomedical engineering 38 (2010): 1188-1203.
74. Abu-Hamdeh, Nidal H., et al. "Controlled drug delivery using the magnetic nanoparticles in non-Newtonian blood vessels." Alexandria Engineering Journal 59.6 (2020): 4049-4062.
75. Luo, Ming, et al. "Micro-/nanorobots at work in active drug delivery." Advanced Functional Materials 28.25 (2018): 1706100.
76. Kuchumov, Alex G., and Aleksander Selyaninov. "Application of computational fluid dynamics in biofluids simulation to solve actual surgery tasks." Human Interaction and Emerging Technologies: Proceedings of the 1st International Conference on Human Interaction and Emerging Technologies (IHIET 2019), August 22-24, 2019, Nice, France. Springer International Publishing, 2020.
77. Botar, C. C., et al. "CFD simulation of the portal vein blood flow." International Conference on Advancements of Medicine and Health Care through Technology: 23–26 September, 2009, Cluj-Napoca, Romania. Springer Berlin Heidelberg, 2009.
78. Shoham, Moshe, et al. "Bone-mounted miniature robot for surgical procedures: Concept and clinical applications." IEEE Transactions on Robotics and Automation 19.5 (2003): 893-901.
79. Zhong, Huiping. "Exploitation and utilization of marine resources and protection of marine ecology." IOP Conference Series: Earth and Environmental Science. Vol. 369. No. 1. IOP Publishing, 2019.
80. Markus, D., et al. "A CFD approach to modeling wave-current interaction." International journal of offshore and polar engineering 23.01 (2013).
81. Markus, D., et al. "A Virtual Free Surface (VFS) model for efficient wave–current CFD simulation of fully submerged structures." Coastal engineering 89 (2014): 85-98.
82. Gundlach, Erich R. "Coastal hazards from oil spills." Coastal Hazards. Dordrecht: Springer Netherlands, 2012. 781-808.
83. Hook, Sharon E. "Beyond thresholds: A holistic approach to impact assessment is needed to enable accurate predictions of environmental risk from oil spills." Integrated Environmental Assessment and Management 16.6 (2020): 813-830.
84. Zhu, Hongjun, Pengzhi Lin, and Qian Pan. "A CFD (computational fluid dynamic) simulation for oil leakage from damaged submarine pipeline." Energy 64 (2014): 887-899.
85. Glatzel, Thomas, et al. "Computational fluid dynamics (CFD) software tools for microfluidic applications–A case study." Computers & Fluids 37.3 (2008): 218-235.
86. Pesch, Simeon, et al. "Rise velocity of live-oil droplets in deep-sea oil spills." Environmental engineering science 35.4 (2018): 289-299.

87. Nakoryakov, Vladimir E., Boris Grigorʹevich Pokusaev, and Isaak Ruvimovich Shreiber. Wave propagation in gas-liquid media. CRC Press, 1993.
88. Kumar, Boddu Sudhir, and Venkaiah Chowdary. "Acoustic performance evaluation of railway boundary walls using a computational fluid dynamics-based simulation approach." Environmental Science and Pollution Research (2024): 1-16.
89. Erbe, Christine. "Effects of underwater noise on marine mammals." The effects of noise on aquatic life. Springer New York, 2012.
90. Cianferra, Marta, and Vincenzo Armenio. "Scaling properties of the Ffowcs-Williams and Hawkings equation for complex acoustic source close to a free surface." Journal of Fluid Mechanics 927 (2021): A2.
91. Kellett, Paula, Osman Turan, and Atilla Incecik. "A study of numerical ship underwater noise prediction." Ocean Engineering 66 (2013): 113-120.
92. Andreini, Antonio, et al. "Development of a turbulent liquid flux model for Eulerian–Eulerian multiphase flow simulations." International Journal of Multiphase Flow 81 (2016): 88-103.
93. Badino, Aglaia, et al. "Normative framework for ship noise: Present and situation and future trends." Noise Control Engineering Journal 60.6 (2012): 740-762.
94. Andreini, Antonio, et al. "Development of a turbulent liquid flux model for Eulerian–Eulerian multiphase flow simulations." International Journal of Multiphase Flow 81 (2016): 88-103.
95. Mushtaq, Muhammad, Nawazish Ali Shah, and Ghulam Muhammad. "Advantages and disadvantages of boundary element methods for compressible fluid flow problems." J. Am. Sci 6.1 (2010): 162-165.
96. Melanson, Mark, Ming Chang, and Wendell Baker II. "Wind tunnel testing's future: a vision of the next generation of wind tunnel test requirements and facilities." 48th AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition. 2010.
97. Klein, Rupert, et al. "Asymptotic adaptive methods for multi-scale problems in fluid mechanics." Journal of Engineering Mathematics 39 (2001): 261-343.
98. Bewley, Thomas R. "Flow control: new challenges for a new renaissance." Progress in Aerospace sciences 37.1 (2001): 21-58.
99. Viswanathan, Hari S., et al. "From fluid flow to coupled processes in fractured rock: Recent advances and new frontiers." Reviews of Geophysics 60.1 (2022): e2021RG000744.
100. Mendez, Miguel Alfonso, et al. "Challenges and opportunities for machine learning in fluid mechanics." arXiv preprint arXiv:2202.12577 (2022).
101. Huang, Y. M., et al. "New Trends in Fluid Mechanics Research." Proceedings of the 5th International Conference on Fluid Mechanics, Shanghai. 2007.