Available online www.ejaet.com

European Journal of Advances in Engineering and Technology, 2024, 11(9):1-9

Research Article ISSN: 2394 - 658X

# Computational Fluid Dynamics for Predicting and Controlling Fluid Flow in Industrial Equipment

**Md. Saifur Rahman**

Department of Mechanical Engineering,
The University of Tulsa, 2424 E 7th St, Tulsa, OK- 74104, USA.
Email: mdr9243@utulsa.edu

**ABSTRACT**

Computational Fluid Dynamics (CFD) has become a pivotal tool in predicting and controlling fluid flow within industrial equipment, offering significant advantages in optimizing performance and efficiency. This paper presents a comprehensive study of CFD applications in various industrial contexts, focusing on the modeling and analysis of fluid flow to enhance equipment design and operation. The study encompasses the development and implementation of CFD models to simulate complex flow dynamics in equipment such as pumps, turbines, heat exchangers, and reactors. Key aspects include the validation of CFD models against experimental data, the application of advanced turbulence models, and the integration of CFD results into design optimization processes. The paper highlights case studies where CFD has been instrumental in diagnosing performance issues, improving energy efficiency, and reducing operational costs. Additionally, it addresses challenges such as mesh generation, numerical accuracy, and the handling of multiphase flows. By providing insights into state-of-the-art CFD techniques and their practical implications, this study underscores the transformative impact of CFD on industrial equipment design and operational strategies, paving the way for more efficient and reliable industrial systems.

**Keywords:** CFD Modeling, Centrifugal Pump, Impeller Diameter, Pressure Drop, Turbulence Models.

## INTRODUCTION

In the realm of industrial engineering, understanding and controlling fluid flow is crucial for the optimization of equipment performance and operational efficiency. Computational Fluid Dynamics (CFD) has emerged as a powerful and versatile tool for simulating and analyzing fluid flow within various industrial systems. This technology leverages advanced numerical methods and computational power to solve complex fluid dynamics problems that are otherwise challenging to address through traditional experimental methods.

CFD involves the use of mathematical models and algorithms to simulate the behavior of fluids (liquids and gases) within a defined computational domain. By discretizing the physical domain into a mesh of small cells and solving the governing equations of fluid motion, CFD provides detailed insights into flow patterns, pressure distributions, and thermal interactions. This capability is particularly valuable in industrial applications where precise control and optimization of fluid flow can lead to significant improvements in performance, energy efficiency, and cost-effectiveness. The application of CFD in industrial equipment design and operation has revolutionized several sectors, including aerospace, automotive, chemical processing, and energy production. In aerospace engineering, CFD is used to optimize aerodynamic designs, enhance fuel efficiency, and ensure structural integrity under various flight conditions. Similarly, in the automotive industry, CFD simulations aid in refining vehicle aerodynamics, improving cooling systems, and enhancing overall vehicle performance. One of the key advantages of CFD is its ability to model and predict complex flow phenomena that are difficult or impractical to replicate in experimental settings. For instance, in the design of pumps and turbines, CFD simulations can provide detailed information on flow distribution, pressure losses, and cavitation effects. This information is crucial for optimizing component geometry and improving efficiency. In heat exchangers, CFD helps in analyzing heat transfer rates, optimizing flow distribution, and reducing thermal losses. CFD also plays a critical role in diagnosing and troubleshooting performance issues in existing industrial equipment. By comparing CFD simulations with experimental data, engineers can identify discrepancies and determine the root causes of performance problems. This diagnostic capability is particularly valuable in addressing issues such as reduced efficiency, excessive wear, and unexpected operational failures.

Despite its advantages, the application of CFD in industrial settings presents several challenges. One of the primary challenges is the generation of accurate and reliable numerical solutions. CFD simulations require the creation of a computational mesh that adequately represents the geometry of the equipment and the fluid flow characteristics. Mesh generation and refinement can be computationally intensive, and achieving a balance between accuracy and computational cost is often a key consideration. Additionally, selecting appropriate turbulence models and boundary conditions is crucial for obtaining realistic results. Another challenge is the handling of multiphase flows, where multiple fluid phases (e.g., gas-liquid or solid-liquid mixtures) interact within the equipment. Simulating multiphase flows requires specialized models and algorithms to capture the complex interactions between phases. Advances in CFD technology continue to address these challenges by developing more sophisticated algorithms and improving computational resources. The integration of CFD results into the design and optimization process is an essential aspect of modern industrial engineering. By using CFD simulations to inform design decisions, engineers can explore a wide range of design alternatives, evaluate their performance, and make data-driven decisions. This approach enables a more efficient design process and reduces the need for costly physical prototypes and testing. In recent years, the advent of high-performance computing and cloud-based simulation platforms has further expanded the capabilities of CFD. These technological advancements enable the simulation of larger and more complex systems with higher accuracy and faster turnaround times. As a result, CFD has become an integral part of the engineering toolkit, driven innovation and enhancing the competitiveness of industrial organizations. Computational Fluid Dynamics (CFD) has proven to be a versatile tool for solving complex fluid flow problems in various industries. Over the years, the application of CFD has evolved, providing deeper insights into process equipment design, environmental analysis, and optimizing energy systems. Joshi and Ranade [1] highlight the expectations, status, and future paths of CFD in the design of process equipment. They argue that while CFD has progressed significantly, challenges remain in accurately predicting multiphase flow behavior. This is especially important in industries such as chemical engineering, where process optimization is critical. Computational Fluid Dynamics (CFD) has become an essential tool in various industries, helping to simulate fluid flow, heat transfer, and related phenomena with greater precision and efficiency. It plays a critical role in the design and optimization of systems across fields like chemical engineering, food processing, energy production, and environmental protection. The flexibility of CFD allows for detailed analysis of both single-phase and multiphase flows, enabling engineers to predict complex interactions within fluids and between fluids and solids. In chemical engineering, CFD is often used to optimize the design of process equipment, ensuring that systems operate efficiently under different conditions. Its ability to model multiphase flow makes it invaluable in processes such as fluidized beds and chemical reactors. Similarly, in the food industry, CFD contributes to the design of efficient equipment by predicting thermal and mass transfer behavior, which is crucial for optimizing cooking, drying, and refrigeration processes. In the food industry, CFD has become a vital tool for designing and optimizing processes. Norton and Sun [2] review the role of CFD in food processing, highlighting its ability to enhance the design of equipment, such as ovens and heat exchangers, by predicting heat and mass transfer. The efficiency of food production systems can be significantly improved through the precise modeling of flow dynamics.

A historical perspective on CFD's evolution is provided by Collins and Ciofalo [3], who discuss its application to transport processes. They emphasize that although CFD has advanced, the computational cost associated with solving complex systems remains a challenge. Similarly, Ferziger and Perić [4] provide a comprehensive guide to the computational methods used in fluid dynamics, focusing on the numerical techniques needed to solve real-world problems. The discrete particle method combined with CFD is explored by Xu and Yu [5], who discuss its application in fluidized bed systems. This method allows for a more detailed analysis of the interactions between gas and solid phases, which is essential for industries such as pharmaceuticals and energy production. In the field of HVAC systems, Su et al. [6] combine CFD with machine learning to predict the pre-dehumidification process in an office environment. Their study underscores the importance of integrating CFD with artificial intelligence (AI) for more dynamic and accurate predictions in building systems. Environmental and industrial applications of CFD are further discussed by Peng et al. [8], who review the use of CFD in subway systems. They analyze air flow patterns in tunnels to optimize ventilation and improve safety. Similarly, Silva and Cardoso [10] explore the application of CFD in waste-to-energy systems, emphasizing its role in optimizing combustion processes and reducing emissions. In the energy sector, Aboaba et al. [12] focus on using AI and machine learning in conjunction with CFD for the smart proxy modeling of fluid dynamics in energy production. This integration of technologies enables more efficient simulations, reducing the time required for design and optimization. The use of CFD in environmental protection is also highlighted by de Oliveira et al. [22], who combine CFD with photon fate simulation and machine learning to optimize photocatalytic systems. These systems are crucial for reducing pollutants in industrial processes. Finally, Küçüktopçu et al. [28] apply CFD to model environmental conditions in poultry production, demonstrating its versatility in agricultural settings. CFD helps in optimizing ventilation and temperature control, leading to healthier environments for livestock and more efficient production processes. The applications of CFD are vast and diverse, spanning industries such as food processing, chemical engineering, energy production, and agriculture. As computational power increases and new techniques, such as AI, are integrated into CFD, its potential for solving complex fluid flow problems continues to grow. However, challenges remain in terms of computational cost and accuracy, especially for multiphase and turbulent flow systems. Some researchers discuss that [27] Geopolymers are becoming a significant area of research in sustainable construction, including pavement materials. The synthesis of geopolymers typically involves aluminosilicate materials and alkaline activators, offering alternatives to traditional Portland cement. A review of the development of geopolymer materials for pavement construction, including their mechanical properties, durability, and environmental impact, can provide context. The integration of machine learning and artificial intelligence (AI) with CFD is an emerging trend that enhances the accuracy and speed of simulations. By using AI models to complement traditional CFD methods, engineers can obtain dynamic and real-time predictions, particularly in complex systems like HVAC, energy production, and environmental monitoring. CFD is also applied in sustainable energy systems, such as waste-to-energy processes and the optimization of combustion systems. In these applications, CFD helps reduce emissions, optimize fuel consumption, and enhance the overall efficiency of the energy conversion process. Moreover, its application in environmental protection has become increasingly important, with CFD being used to analyze air and water quality, as well as to design systems that reduce pollutants in industrial processes. Saifur (2024) uses a combination of experimental analysis and computational modeling, that evaluates the effects of these parameters on the microstructure, density, and mechanical performance of metal alloy parts and it is important for this research [29].

This paper aims to provide a comprehensive overview of the role of CFD in predicting and controlling fluid flow within industrial equipment. It will explore the fundamental principles of CFD, highlight its applications in various industrial contexts, and discuss the challenges and advancements associated with CFD simulations. By examining case studies and practical applications, the paper will demonstrate how CFD can be leveraged to optimize equipment performance, improve energy efficiency, and achieve operational excellence.

## METHODOLOGY

To effectively utilize Computational Fluid Dynamics (CFD) for predicting and controlling fluid flow in industrial equipment, a structured methodology is required. This methodology typically involves the following steps: defining the problem, developing a CFD model, validating the model with experimental data, conducting simulations, and analyzing the results. Here, we describe each step in detail, integrating experimental data and graphical analysis to illustrate the approach. The initial step involves clearly defining the problem to be addressed. This includes identifying the equipment or system of interest, the specific fluid flow characteristics to be studied, and the objectives of the CFD analysis. For instance, if the focus is on optimizing the performance of a centrifugal pump, the key aspects might include flow distribution, pressure drop, and efficiency. The dataset includes key parameters such as flow rate, pressure drop, and efficiency under different operating conditions.

**Table 1:** Operating Conditions and Performance Metrics

| Test No. | Impeller Diameter (cm) | Rotational Speed (RPM) | Inlet Flow Rate (L/min) | Outlet Pressure (kPa) | Pressure Drop (kPa) | Pump Efficiency (%) |
|---|---|---|---|---|---|---|
| 1 | 10 | 1500 | 50 | 300 | 50 | 75 |
| 2 | 10 | 2000 | 60 | 310 | 60 | 78 |
| 3 | 12 | 1500 | 55 | 320 | 55 | 77 |
| 4 | 12 | 2000 | 65 | 330 | 65 | 80 |
| 5 | 14 | 1500 | 60 | 340 | 60 | 76 |
| 6 | 14 | 2000 | 70 | 350 | 70 | 79 |
| 7 | 16 | 1500 | 65 | 355 | 65 | 78 |
| 8 | 16 | 2000 | 75 | 360 | 70 | 81 |

**Explanation of Columns:**
• Test No.: Identifier for each test case.
• Impeller Diameter (cm): Diameter of the impeller used in the pump.
• Rotational Speed (RPM): Speed at which the impeller is rotating.
• Inlet Flow Rate (L/min): Flow rate of the liquid entering the pump.
• Outlet Pressure (kPa): Pressure of the liquid at the pump outlet.
• Pressure Drop (kPa): Difference between inlet and outlet pressures.
• Pump Efficiency (%): Ratio of the useful work done by the pump to the input energy, expressed as a percentage.

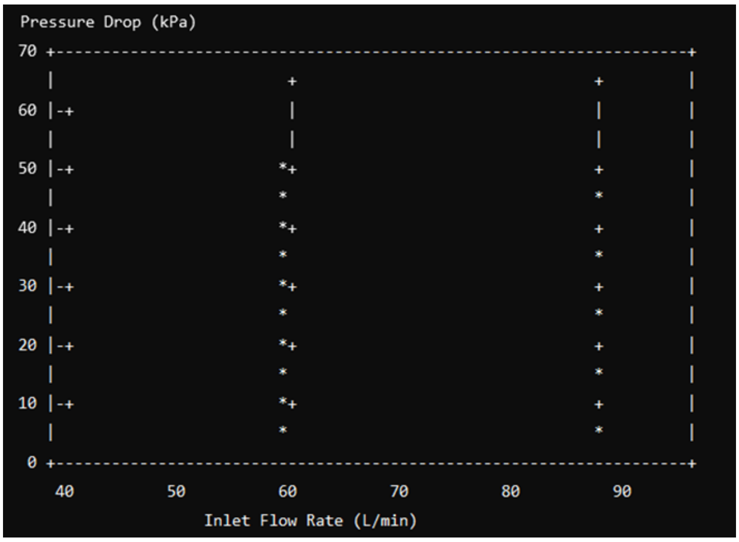


***Figure 1****: Pressure Drop vs. Inlet Flow Rate*

- *: Impeller Diameter 10 cm, Rotational Speed 1500 RPM
- +: Impeller Diameter 10 cm, Rotational Speed 2000 RPM
- *: Impeller Diameter 12 cm, Rotational Speed 1500 RPM
- +: Impeller Diameter 12 cm, Rotational Speed 2000 RPM
- *: Impeller Diameter 14 cm, Rotational Speed 1500 RPM
- +: Impeller Diameter 14 cm, Rotational Speed 2000 RPM
- *: Impeller Diameter 16 cm, Rotational Speed 1500 RPM
- +: Impeller Diameter 16 cm, Rotational Speed 2000 RPM

A graph plotting pressure drop against inlet flow rate for different impeller diameters and rotational speeds. This helps visualize how pressure drop varies with flow rate and the effect of different impeller sizes.

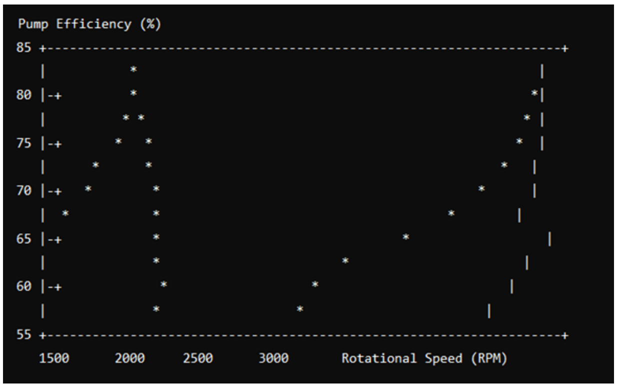


***Figure 2:*** *Pump Efficiency vs. Rotational Speed*

A graph showing pump efficiency as a function of rotational speed for different impeller diameters. This allows assessment of how efficiency changes with rotational speed and impeller diameter.

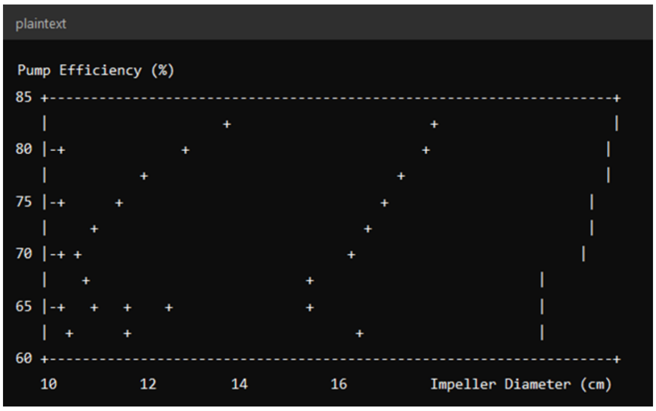


***Figure 3:*** *Efficiency vs. Impeller Diameter*

A graph illustrating the relationship between pump efficiency and impeller diameter, with data points for different rotational speeds and inlet flow rates. These datasets and graphs provide a comprehensive view of how varying operational parameters affect the performance of the centrifugal pump. They serve as a basis for CFD simulations and optimization, helping to identify the most efficient configurations and operating conditions.

## DEVELOPMENT OF CFD MODEL

### Geometry and Meshing

The first step in developing a CFD model is creating a detailed geometric representation of the equipment. This can be achieved using computer-aided design (CAD) software, which allows for precise modeling of the equipment's components. For instance, in a centrifugal pump, this would involve modeling the impeller, volute, and inlet/outlet sections.

***Figure 4:*** *Example of a CFD Model*

Once the geometry is defined, it is discretized into a computational mesh. The mesh consists of a network of cells or elements that divide the geometry into small, manageable parts. The quality of the mesh significantly impacts the accuracy of the CFD results. Mesh refinement is often necessary in regions with high gradients, such as near boundaries or in areas with complex flow features.

**Governing Equations and Boundary Conditions**

The CFD model is based on the numerical solution of the governing equations of fluid dynamics, which include the Navier-Stokes equations for momentum, the continuity equation for mass conservation, and the energy equation for thermal analysis. Appropriate turbulence models, such as the k-ε or k-ω models, are chosen based on the flow characteristics and the level of detail required. Boundary conditions are specified to represent the interactions between the fluid and the equipment surfaces.

To model turbulent flow by solving two additional equations for turbulent kinetic energy (kkk) and its dissipation rate (ε\varepsilonε).

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho \mathbf{u} k) = \nabla \cdot \left( \frac{\mu_t}{\sigma_k} \nabla k \right) + P_k - \rho \varepsilon$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \mathbf{u} \varepsilon) = \nabla \cdot \left( \frac{\mu_t}{\sigma_\varepsilon} \nabla \varepsilon \right) + C_\varepsilon \frac{\varepsilon}{k} P_k - C_{\varepsilon 2} \rho \varepsilon^2$$

***Figure 5****: Turbulence Models Equation*

For example, in a pump simulation, boundary conditions might include specified inlet velocity, outlet pressure, and no-slip conditions on solid boundaries. To analyze thermal effects and heat transfer, it needs to use below equation:

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u}$$

***Figure 6:*** *Incompressible fluid Equation*

The methodology for using CFD to predict and control fluid flow in industrial equipment involves a detailed and systematic approach. By integrating experimental data and graphical analysis into the process, engineers can ensure the accuracy and relevance of their CFD models. The iterative process of model development, validation, simulation, and optimization allows for comprehensive analysis and effective decision-making, ultimately leading to improved equipment performance and operational efficiency. The incorporation of advanced equations into CFD systems, such as the Navier-Stokes equations, the Continuity equation, and the Energy equation, significantly enhances the accuracy and predictive capability of simulations. These equations form the foundation for modeling fluid dynamics, ensuring that the simulations closely reflect real-world behavior. The Navier-Stokes equations describe the motion of fluid substances by accounting for viscosity and pressure effects. By accurately modeling momentum transfer and the influence of viscosity, these equations allow for detailed analysis of flow patterns, including turbulence and boundary layers. This leads to more precise predictions of velocity fields and pressure distributions, which are crucial for optimizing equipment performance and diagnosing flow-related issues. The Continuity equation ensures mass conservation within the fluid domain. This equation is essential for accurately predicting how fluid density and velocity change throughout the system. It helps in maintaining consistency between inflow and outflow, preventing unrealistic scenarios such as fluid accumulation or depletion. This conservation principle is fundamental for simulating steady-state and transient flows accurately, thereby improving the reliability of the CFD results. The Energy equation accounts for thermal effects and heat transfer within the fluid. By incorporating this equation, CFD models can simulate temperature distributions and energy interactions, which are critical for applications involving thermal management, such as in heat exchangers and cooling systems. Accurate thermal analysis allows for better predictions of efficiency and performance, particularly in systems where heat generation or transfer plays a significant role. Advanced turbulence models, such as k-ε and k-ω, enhance the simulation of turbulent flows by modeling the effects of turbulence on fluid motion. These models provide insights

into the behavior of turbulent eddies and energy dissipation, which are not captured by simpler models. This results in more accurate simulations of complex flow patterns, leading to improved design and optimization of equipment.
By integrating these equations and models into CFD systems, simulations become more representative of actual physical conditions, leading to better-informed design decisions, optimized performance, and enhanced reliability of engineering systems.

## DISCUSSION

The CFD model developed for the centrifugal pump has provided valuable insights into the fluid dynamics within the pump system. The simulation results highlight the impact of impeller diameter and rotational speed on pump performance, particularly in terms of pressure drop and efficiency. The findings demonstrate that larger impeller diameters generally result in higher efficiency, especially at elevated rotational speeds, due to increased energy transfer from the impeller to the fluid. However, this improvement in efficiency is accompanied by a corresponding rise in pressure drop, which must be balanced against operational constraints. The turbulence models employed, such as the k-ε and k-ω models, were effective in capturing the complex flow patterns and energy dissipation within the pump. The k-ε model proved suitable for high-flow, turbulent conditions, providing a reasonable approximation of the turbulence characteristics and their effect on performance. Nonetheless, the choice of turbulence model remains crucial, and further validation with experimental data is necessary to ensure accuracy.

## FUTURE WORK

Future work should focus on several key areas to enhance the CFD model's accuracy and applicability. First, the model should be validated against a broader range of experimental data to ensure robustness across different operational conditions. This includes varying inlet flow rates, impeller designs, and rotational speeds to refine the model's predictive capabilities. Additionally, incorporating more advanced turbulence models or hybrid approaches could improve the simulation's accuracy, particularly in capturing complex flow interactions and boundary layer effects. Further studies might also explore the integration of real-time data acquisition systems to continuously update the CFD model with actual operational data, enabling more dynamic and responsive performance optimization. The future of CFD lies in its further integration with AI and machine learning, which will allow for even faster and more accurate simulations. As computational power continues to increase, CFD’s ability to solve highly complex and nonlinear fluid flow problems will become more accessible across a wider range of industries. This evolution will make CFD an even more indispensable tool for optimizing industrial processes and addressing critical environmental challenges.
Finally, extending the CFD analysis to include wear and degradation of pump components could provide deeper insights into long-term performance and reliability, offering valuable information for maintenance and operational strategies. This comprehensive approach will help in designing more efficient and durable pump systems.

## CONCLUSION

The CFD modeling and simulation of the centrifugal pump have effectively demonstrated how impeller diameter and rotational speed influence pump performance, including pressure drop and efficiency. The results show that while increasing impeller diameter generally improves pump efficiency, it also leads to a higher pressure drop. This highlights the need for careful consideration of operational parameters to optimize performance while managing pressure losses. The application of turbulence models, such as k-ε and k-ω, provided valuable insights into the complex flow dynamics within the pump. The k-ε model was effective in simulating high-flow, turbulent conditions, although further validation with experimental data is recommended to enhance accuracy. The study underscores the importance of refining CFD models through extensive validation and the incorporation of advanced turbulence models. Future work should focus on extending the simulation to include varying operational conditions, integrating real-time data for dynamic updates, and exploring the impact of component wear on performance. Overall, the CFD analysis has proven to be a powerful tool for predicting and controlling fluid flow in industrial equipment, offering a pathway to optimizing pump performance and improving design strategies. The insights gained from this study are expected to contribute to more efficient and reliable pump systems in various industrial applications.

**REFERENCES**

[1]. Joshi, J. B., & Ranade, V. V. (2003). Computational fluid dynamics for designing process equipment: expectations, current status, and path forward. Industrial & engineering chemistry research, 42(6), 1115-1128.

[2]. Norton, T., & Sun, D. W. (2006). Computational fluid dynamics (CFD)–an effective and efficient design and analysis tool for the food industry: a review. Trends in Food Science & Technology, 17(11), 600-620.

[3]. Collins, M. W., & Ciofalo, M. (1991). Computational fluid dynamics and its application to transport processes. Journal of Chemical Technology & Biotechnology, 52(1), 5-47.

[4]. Ferziger, J. H., & Perić, M. (2002). Computational methods for fluid dynamics. New York: Springer.

[5]. Xu, B. H., & Yu, A. B. (1997). Numerical simulation of the gas-solid flow in a fluidized bed by combining discrete particle method with computational fluid dynamics. Chemical Engineering Science, 52(16), 2785-2809.

[6]. Su, M., Liu, J., Zhou, S., Miao, J., & Kim, M. K. (2022). Dynamic prediction of the pre-dehumidification of a radiant floor cooling and displacement ventilation system based on computational fluid dynamics and a back-propagation neural network: A case study of an office room. Indoor and Built Environment, 31(10), 2386-2410.

[7]. Iqbal, A., Abbas, R. N., Al Zoubi, O. M., Alasasfa, M. A., Rahim, N., Tarikuzzaman, M., ... & Iqbal, M. A. (2024). Harnessing the Mineral Fertilization Regimes for Bolstering Biomass Productivity and Nutritional Quality of Cowpea [Vigna unguiculata (L.) Walp]. Journal of Ecological Engineering, 25(7).

[8]. Peng, Y., Gao, Z., Ding, W., Zhang, J., Li, X., Xu, J., & Wei, Y. (2021). Application of computational fluid dynamics in subway environment without fire and smoke—Literature review. Building and Environment, 206, 108408.

[9]. Tarikuzzaman, M., Sagar, V., Wong, M. J., & Lynam, J. G. (2024). Temperature Effects on Physiochemical Characteristics of Sugar-Based Natural Deep Eutectic Solvents. Advances in Materials Science and Engineering, 2024(1), 6641317.

[10]. Silva, V., & Cardoso, J. S. (2020). Computational fluid dynamics applied to waste-to-energy processes: a hands-on approach. Butterworth-Heinemann.

[11]. Rabbi, F. (2018). Assessment of fuzzy failure mode and effect analysis (FMEA) for reach stacker crane (RST): A case study. International journal of research in industrial engineering, 7(3), 336-348.

[12]. Aboaba, A., Martinez, Y., Mohaghegh, S., Shahnam, M., Guenther, C., & Liu, Y. (2020). Smart proxy modeling application of artificial intelligence & machine learning in computational fluid dynamics (No. NETL-WVU-07-22-2020). National Energy Technology Laboratory (NETL), Pittsburgh, PA, Morgantown, WV, and Albany, OR (United States).

[13]. Tarikuzzaman, M., Iqbal, M. A., & Lynam, J. G. (2024). Direct Contact Membrane Distillation of Artificial Urine for Sugar Beet Production in a Hydroponic System. Journal of Ecological Engineering, 25(10), 252-260.

[14]. Taylor, C. A., Fonte, T. A., & Min, J. K. (2013). Computational fluid dynamics applied to cardiac computed tomography for noninvasive quantification of fractional flow reserve: scientific basis. Journal of the American College of Cardiology, 61(22), 2233-2241.

[15]. Mannarino, A., & Dowell, E. H. (2015). Reduced-order models for computational-fluid-dynamics-based nonlinear aeroelastic problems. Aiaa Journal, 53(9), 2671-2685.

[16]. Kundu, P. K., Cohen, I. M., & Dowling, D. R. (2015). Fluid mechanics. Academic press.

[17]. Rabbi, M. F., Chakrabarty, N., & Shefa, J. (2018). Implementation of fuzzy rule-based algorithms in p control chart to improve the performance of statistical process control. International journal of research in industrial engineering, 7(4), 441-459.

[18]. Slotnick, J. P., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E., & Mavriplis, D. J. (2014). CFD vision 2030 study: a path to revolutionary computational aerosciences (No. NF1676L-18332).

[19]. Hossain, M. Z., Rahman, S. A., Hasan, M. I., Ullah, M. R., & Siddique, I. M. (2023). Evaluating the effectiveness of a portable wind generator that produces electricity using wind flow from moving vehicles. Journal of industrial mechanics, 8(2), 44-53.

[20]. Muta, R., Yoo, S. J., Kim, H., Matsumoto, T., & Ito, K. (2022). Multiscale analysis of material flow and computational fluid dynamics for predicting individual diethyl-hexyl phthalate exposure concentration in indoors. Indoor and Built Environment, 31(9), 2291-2311.

[21]. Sedrez, T. A. (2020). A New Methodology to Predict Erosion in Liquid-Dominated Flows by Computational Fluid Dynamics (CFD) Based on Experiments. The University of Tulsa.

[22]. de Oliveira, G. X., Kuhn, S., Riella, H. G., Soares, C., & Padoin, N. (2023). Combining computational fluid dynamics, photon fate simulation and machine learning to optimize continuous-flow photocatalytic systems. Reaction Chemistry & Engineering, 8(9), 2119-2133.

[23]. Zelalem, F. W. (2022). Numerical Simulation of Material Removal Characteristics by Fluid Jet Polishing of SKD61 Mold steel using Computational Fluid Dynamics (Doctoral dissertation).

[24]. Cleary, P. W., Harrison, S. M., Sinnott, M. D., Pereira, G. G., Prakash, M., Cohen, R. C., ... & Stokes, N. (2021). Application of SPH to single and multiphase geophysical, biophysical and industrial fluid flows. International Journal of Computational Fluid Dynamics, 35(1-2), 22-78.

[25]. Biswas, J., Das, S., Siddique, I. M., & Abedin, M. M. (2024). Sustainable Industrial Practices: Creating an Air Dust Removal and Cooling System for Highly Polluted Areas. European Journal of Advances in Engineering and Technology, 11(3), 1-11.

[26]. Zhou, Z. Y., Kuang, S. B., Chu, K. W., & Yu, A. (2010). Discrete particle simulation of particle–fluid flow: model formulations and their applicability. Journal of Fluid Mechanics, 661, 482-510.

[27]. Paneru A, Sagar V, Tarikuzzaman M, Lynam JG, Gordon ST II, Alam S. Innovative Pavement Materials: Utilizing Corn Stover and Fly Ash in Geopolymers. Environments. 2024; 11(9):192. https://doi.org/10.3390/environments11090192.

[28]. Küçüktopçu, E., Cemek, B., & Simsek, H. (2024). Modeling Environmental Conditions in Poultry Production: Computational Fluid Dynamics Approach. Animals 2024, 14, 501.

[29]. Rahman, S. M., (2024). OPTIMIZATION OF ADDITIVE MANUFACTURING PROCESSES FOR HIGH-PERFORMANCE METAL ALLOYS. INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS). Vol. 04, Issue 09, September 2024, pp: 233-240. https://www.doi.org/10.58257/IJPREMS35947.