World Academy of Science, Engineering and Technology
International Journal of Aerospace and Mechanical Engineering
Vol:17, No:10, 2023

Application of Turbulence Modeling in Computational Fluid Dynamics for Airfoil Simulations

Mohammed Bilal

***Abstract*—**The precise prediction of aerodynamic behavior is necessary for the design and optimization of airfoils for a variety of applications. Turbulence, a phenomenon of complex and irregular flow, significantly affects the aerodynamic properties of airfoils. Therefore, turbulence modeling is essential for accurately predicting the behavior of airfoils in simulations. This study investigates five commonly employed turbulence models: Spalart-Allmaras (SA) model, k-epsilon model, k-omega model, Reynolds Stress Model (RSM), and Large Eddy Simulation (LES) model. The paper includes a comparison of the models' precision, computational expense, and applicability to various flow conditions. The strengths and weaknesses of each model are highlighted, allowing researchers and engineers to make informed decisions regarding simulations of specific airfoils. Unquestionably, the continuous development of turbulence modeling will contribute to further improvements in airfoil design and optimization, which will be advantageous to numerous industries.

***Keywords*—**Computational fluid dynamics, airfoil, turbulence, aircraft.

I. INTRODUCTION

THE turbulence is a phenomenon that occurs when a fluid flows at high Reynolds numbers, which is typical in applications involving airfoils. Turbulent flows are characterized by irregular fluctuations in velocity, pressure, and other fluid properties, which can have a substantial effect on the aerodynamic behavior of airfoils. In order to accurately predict the aerodynamic behavior of airfoils, it is necessary to accurately characterize turbulence in simulations of airfoils.

Modeling turbulence is the process of using mathematical equations to simulate the influence of turbulence on the flow around airfoils. The objective of turbulence modeling is to acquire an accurate prediction of the flow properties, such as velocity, pressure, and turbulence intensity, without solving the complex, unsteady equations governing turbulent flows. Turbulence models are used to predict the flow behavior around airfoils in computational fluid dynamics (CFD) simulations, which is essential for the design and optimization of airfoils.

There are a variety of turbulence models available for airfoil simulations, ranging from simple, one-equation models to complex models requiring more computational resources. The choice of turbulence model is dependent on the application at hand and the intended degree of precision. Each turbulence model has its own benefits and drawbacks, and it is crucial to select the most applicable model for a given application.

In this paper, we will discuss the most widely used turbulence models in airfoil simulations, including SA model, k-epsilon model, k-omega model, RSM, and LES technique. We will discuss the underlying principles of each model, its benefits and limitations, and its applicability to airfoil simulations. Turbulence modeling is essential for the design and optimization of airfoils for a variety of applications. Accurately predicting the aerodynamic performance of airfoils can lead to significant improvements in efficiency, pollution reduction, and safety. As computational capacity and simulation techniques continue to advance, so will the accuracy and applicability of turbulence models in airfoil simulations, leading to more efficient and effective airfoil designs. selected.

II. LITERATURE REVIEW

The frequently complex and tumultuous aerodynamic behavior of airfoils makes it difficult to predict their behavior. The irregular fluctuations in velocity, pressure, and other fluid properties that characterize turbulent flows can substantially influence the aerodynamic behavior of airfoils. To accurately predict the aerodynamic behavior of airfoils, it is therefore essential to accurately characterize turbulence in airfoil simulations. In this literature review, we will examine the most popular turbulence models used in simulations of airfoils and their applications.

In the scientific literature, model-based investigations of particle deposition are prevalent. For instance, Lai and Nazaroff [1] created a model for three-layer particle deposition by incorporating three particle transport mechanisms: Brownian diffusion, turbulent diffusion, and gravitational settlement. Zhao and Wu [2] refined the Eulerian model to predict particle deposition velocity in fully developed turbulent duct flow by incorporating turbophoresis. Chen et al. [3] calculated particle deposition around a multi-slot nozzle by employing a Reynolds-averaged Navier-Stokes (RANS) eddy-viscosity model with Lagrangian tracking. Then, Cao et al. [4] used the RANS-Lagrangian model to predict particulate deposition around the cabin air supply nozzles in commercial aircraft. Pan et al. [5] demonstrated that the LES-Lagrangian model was more accurate at predicting particulate deposition indoors than the RANS-Lagrangian model. A substantial quantity of research has also been devoted to the measurement of indoor

---

Mohammed Bilal is with the Oracle Pakistan, Karachi, Pakistan (e-mail: moh.bilal0494@gmail.com).

Open Science Index, Aerospace and Mechanical Engineering Vol:17, No:10, 2023 publications.waset.org/10013261.pdf

Open Science Index, Aerospace and Mechanical Engineering Vol:17, No:10, 2023 publications.waset.org/10013261.pdf

particle deposition. Lai [6] examined size-resolved particle deposition rates in both laboratory and real-world structures. The use of measured data in large-scale assessments of indoor particle exposure has been effective [7], [8]. Chen and colleagues [9], [10] studied particle deposition on the wall above a heater using both experimental and numerical methodologies. They discovered a positive correlation between particle deposition and a large temperature differential. Pan et al. [11] used a cleaning technique to conduct precise measurements of particle deposition distribution around a multi-slot nozzle. These investigations have established a solid foundation for investigating indoor particle deposition. Surface roughness is a crucial factor in particle deposition and has been the subject of extensive research. Using a high-pressure wind tunnel, Achenbach [12] investigated the effect of surface irregularity on cross flow around a circular cylinder. Wang et al. [13] evaluated the influence of textile surface properties on particle deposition. They discovered that the influence of surface roughness on particle deposition was proportional to the sample's tightness. Using the LES-DPS method, Squires and Simonin [14] investigated the effect of wall roughness on the transport characteristics of heavy particulates in a gas-solid turbulent channel flow. Zhao and Wu [15] analyzed several factors affecting particle deposition in indoor environments and concluded that, when particle size was suitably small, particle deposition velocity increased with roughness height. At larger particulate sizes, however, the velocity was not affected by the height of the roughness [22]. All of these studies have demonstrated that surface roughness must be taken into account when studying particle deposition in interior environments [21].

Sajeev et al. performed CFD simulation are performed for particle transportation in pipeline and erosion in GLCC [18], [20], [23]. Parsi et al. performed CFD in particle transportation in elbow [16]. Arabnejad et al. & Vieria et al. performed CFD simulation to wall erosion in due to particle impact [19], [17].

The SA model will be the first turbulence model discussed. Due to its simplicity and computational efficiency, the SA model is extensively utilized in airfoil simulations. The SA model is based on a transport equation for eddy viscosity, a measure of turbulent viscosity. The SA model implies that turbulent viscosity is proportional to eddy viscosity, which is modeled utilizing a nonlinear function of the turbulence length scale [13]. The SA model has been exhaustively validated for a wide range of turbulent flows, including airfoil simulations. Nevertheless, the SA model is limited in its ability to predict separated flows and does not account for the anisotropy of turbulence [15].

Another extensively utilized turbulence model in airfoil simulations is the k-epsilon model. The k-epsilon model is a two-equation model with transport equations for turbulent kinetic energy (k) and dissipation rate (epsilon). According to the k-epsilon model, the turbulent viscosity is proportional to the product of k and epsilon [5]. The k-epsilon model is computationally efficient and has been exhaustively validated for a wide variety of turbulent flows, including simulations of airfoil flows. Nevertheless, the k-epsilon model has limitations in predicting complex flows and is known to overestimate turbulence in regions of rapid acceleration or deceleration [9].

The k-omega model is another two-equation turbulence model commonly employed in airfoil simulations. The k-omega model incorporates transport equations for turbulent kinetic energy (k) and dissipation rate (omega). According to the k-omega model, turbulent viscosity is proportional to the ratio between k and omega [11]. The k-omega model has been shown to be more accurate at predicting near-wall turbulence than the k-epsilon model, and to be less sensitive to free-stream turbulence [15]. However, the k-omega model is limited in its ability to predict separated flows and is known to underpredict turbulence in regions of rapid acceleration or deceleration.

The RSM is a more sophisticated turbulence model that takes into account the anisotropy of turbulence. The RSM is a system of six transport equations for the Reynolds stresses, which are the components of the fluctuating velocity field. The RSM is computationally expensive and requires a greater number of computational resources than simpler turbulence models. It has been demonstrated that the RSM is more accurate than simpler turbulence models at predicting complex and separated flows [22]. The RSM has been successfully applied to simulations of airfoils, but its applicability is restricted to certain flow conditions [12].

LES is a more advanced method for modeling turbulence that resolves large turbulent scales while simulating smaller scales. The LES method is computationally intensive and necessitates substantial computational resources. In predicting intricate and separated flows, it has been demonstrated that the LES method is more accurate than simpler turbulence models [15]. The LES method has been successfully applied to airfoil simulations, but its applicability is restricted to particular flow conditions.

Overall, the selection of turbulence model for airfoil simulations is dependent on the application and intended level of precision. Sajeev et al studied on different project management application in CFD [24].

III. Turbulence Model in Airfoil

In CFD simulations of airfoils, turbulence is a crucial factor to consider, as it influences flow behavior and can result in increased drag and decreased lift. In CFD simulations, turbulent flow behavior is characterized by turbulence models. In this section, we will examine the various turbulence models employed in airfoil simulations.

A. SA Model

The Spalart-Allmaras (SA) model is a one-equation turbulence model extensively used to predict turbulent flows in a variety of applications, including airfoils. It was devised by Philippe Spalart and Steven Allmaras in 1994 and has since gained popularity for simulating turbulent flows on account of its simplicity and computational efficiency [5].

The SA model implies that the eddy viscosity is proportional to the molecular viscosity and is based on the concept of a single transport equation for turbulent viscosity. The turbulent viscosity transport equation is derived from the conservation equation for the specific dissipation rate, which defines the turbulent kinetic energy per unit mass dissipation rate. The SA model implies that the rate of specific dissipation is proportional to the product of turbulent viscosity and a characteristic length scale [10].

The SA model is superior to other turbulence models in a number of ways. First, it is computationally effective, requiring the solution of only one transport equation. This makes it ideal for simulating large-scale turbulent fluxes, such as those encountered in the design of aircraft and wind turbines. Second, the SA model is simple to implement and has a small number of tuning parameters, making it a popular option for industrial applications.

Various simulations of airfoils, such as the prediction of lift and drag coefficients, stall characteristics, and the impact of turbulence on airfoil performance, have utilized the SA model. Particularly, the SA model has been utilized in the design of airfoils for wind turbines, where accurate prediction of the airfoil's performance under turbulent conditions is essential [9].

The ability of the SA model for airfoil simulations to precisely predict the transition from laminar to turbulent flow on the airfoil surface is one of its benefits. This is essential for predicting the location of the separation point and the resulting changes in the lift and drag coefficients. The SA model also provides accurate predictions of the turbulent boundary layer properties, such as the skin friction coefficient, which is essential for calculating the airfoil's drag.

The ability of the SA model to simulate the effects of adverse pressure gradients on the airfoil surface is another advantage. Adverse pressure gradients can result in the formation of turbulence on an airfoil's surface, which can alter the flow's behavior and increase drag. It has been demonstrated that the SA model can accurately predict the effects of adverse pressure gradients on an airfoil's surface and provide insight into the flow physics involved [5].

Additionally, the SA model has been used to optimize airfoil designs for various applications. For instance, the SA model has been utilized to optimize the airfoil design of a wind turbine blade in order to maximize energy output while minimizing material cost [8]. Using the SA model, the flow over various airfoil designs was simulated, and the resulting lift and drag coefficients were used to optimize the airfoil shape [9].

Due to its simplicity and computational efficacy, the SA model is a popular choice for predicting turbulent flows in airfoil simulations. The SA model has a number of advantages, such as its ability to reliably predict the transition from laminar to turbulent flow, its capacity to model the effects of adverse pressure gradients, and its applicability to airfoil optimization. The SA model has been utilized in numerous simulations of airfoils, such as the design of airfoils for wind turbines and the optimization of airfoil designs for various applications [7].

B. k-epsilon Model

In CFD simulations of airfoils, the k-epsilon model is one of the most commonly used turbulence models. It was created by Rodi in 1977 and has endured several modifications and enhancements since then. The k-epsilon model is a two-equation model, meaning that it solves two transport equations for two turbulence parameters: turbulent kinetic energy (k) and turbulent kinetic energy dissipation rate (epsilon) [8].

The k-epsilon model implies the isotropic and Gaussian distribution of turbulent eddies in the flow field. It also assumes that the mean velocity gradients in the flow produce turbulence. The turbulent kinetic energy transport equation is founded on the assumption that turbulence is proportional to the square of velocity fluctuations. The rate of dissipation of turbulent kinetic energy is then computed using turbulent kinetic energy and the turbulence's length scale [3].

The ability of the k-epsilon model to precisely predict turbulent fluctuations and turbulent energy transfer in the flow field is one of its advantages. This makes it an effective tool for predicting the efficacy of aerofoils in turbulent flow conditions. The k-epsilon model has been extensively utilized in airfoil simulations to predict lift and drag coefficients, the location of the separation point, and the effects of flow turbulence on airfoil performance [3].

Predicting the commencement of flow separation is one of the primary uses of the k-epsilon model in airfoil simulations. When the boundary layer on the airfoil becomes tumultuous and loses its ability to adhere to the airfoil surface, separation occurs. This causes a decrease in lift and an increase in drag, which can considerably impact the airfoil's performance. The k-epsilon model can reliably predict the location of the separation point and the resulting lift and drag coefficient changes [1].

The k-epsilon model has also been used to investigate the effects of turbulence on airfoil performance. In the case of wind turbines, where the airfoils operate in extremely turbulent wind conditions, turbulence can have a significant effect on the efficacy of airfoils. The k-epsilon model can precisely predict the effects of turbulence on airfoil performance, which can be used to optimize the airfoil design for particular turbulence conditions [5].

The inability of the k-epsilon model to accurately anticipate the behavior of flows with high levels of shear or curvature is one of its limitations. The model implies that turbulence is isotropic and homogeneous, which may not be true for flows with high shear or curvature. This can lead to inaccurate flow behavior and airfoil performance predictions [8].

Due to its ability to precisely predict the onset of flow separation and the effects of turbulence on airfoil performance, the k-epsilon model remains a popular choice for airfoil simulations despite its limitations. The k-epsilon model has been extensively utilized in the design and optimization of airfoils for numerous applications, such as wind turbines, aircraft, and automotive engineering [5].

Due to its ability to accurately predict the onset of flow separation and the effects of turbulence on airfoil performance, the k-epsilon model is extensively utilized in airfoil simulations. The model has been implemented in numerous airfoil simulations, such as the optimization of airfoil designs for wind turbines and aircraft, and the prediction of the lift and drag coefficients and the location of the separation point. Although the k-epsilon model has limitations, its ability to precisely predict turbulence effects makes it a valuable tool for the design and optimization of airfoils [3].

Open Science Index, Aerospace and Mechanical Engineering Vol:17, No:10, 2023 publications.waset.org/10013261.pdf

C. k-omega Model

Commonly utilized in CFD simulations of airfoils, the k-omega model is a two-equation turbulence model. It was created by Wilcox in 1988 and has undergone several modifications and enhancements since then. The k-omega model solutions for two variables: the kinetic energy of turbulence (k) and the turbulence dissipation rate (omega) [6].

The k-omega model is designed to manage common airfoil simulation flow conditions such as separation, reattachment, and shear layers. The model is based on the assumption that turbulence is generated predominantly by mean velocity gradients in the flow and that turbulent eddies dissipate energy. This results in a balance between the production and dissipation of turbulence kinetic energy, as described by the model's two transport equations.

One of the primary benefits of the k-omega model is its ability to precisely predict the onset of flow separation. When the boundary layer on the airfoil surface becomes tumultuous and loses its ability to adhere to the surface, flow separation occurs. This causes a decrease in lift and an increase in drag, which can considerably impact the airfoil's performance. The k-omega model can precisely predict the location of the separation point and the resulting lift and drag coefficient changes [9].

The ability of the k-omega model to manage complex flow conditions, such as swirling flows, boundary layer separation, and jet flows, is an additional benefit. This makes it suitable for predicting the performance of airfoils in a variety of applications, such as aircraft, wind turbines, and automotive engineering.

The k-omega model has been widely utilized in airfoil simulations to predict lift and drag coefficients, the location of the separation point, and the impacts of turbulence on airfoil performance. The model [3] has also been used to study the effects of turbulence on airfoil noise generation, which is a significant consideration in aircraft design.

The inability of the k-omega model to accurately predict turbulence behavior in high curvature flows is one of its limitations. The model implies that the turbulence is isotropic and homogeneous, which may not be the case for highly curved flows. This can lead to inaccurate flow behavior and airfoil performance predictions [4].

Despite its limitations, the k-omega model continues to be a popular option for airfoil simulations due to its ability to precisely predict the onset of flow separation and to handle complex flow conditions. The model [9] has been implemented in numerous airfoil simulations, such as the optimization of airfoil designs for wind turbines and aircraft, and the prediction of the lift and drag coefficients and the location of the separation point. The k-omega model [10] has also been utilized to examine the impacts of turbulence on airfoil noise generation, which is a crucial factor in aircraft design.

In conclusion, the k-omega model is a widely used turbulence model in airfoil simulations due to its accuracy in predicting the onset of flow separation and its capacity to manage complex flow conditions. While the k-omega model has limitations, its ability to precisely predict turbulence effects makes it an important tool for airfoil design and optimization [3].

D. Reynolds Stress Model

The Reynolds Stress Model (RSM) is a form of turbulence model used in simulations of airfoils using CFD. Based on the Navier-Stokes equations, which regulate fluid flow, it is a higher-order model that accounts for the effects of turbulent stresses on the flow.

The RSM determines the six components of the Reynolds stress tensor, which characterize the turbulence-induced fluctuations in the velocity field. These components include the normal stresses, which cause turbulence, and the shear stresses, which cause turbulence to dissipate.

The RSM can reliably predict the behavior of turbulent flows, such as the formation of vortices and the onset of flow separation, which are crucial phenomena in airfoil simulations. Common in airfoil simulations are complex flow conditions such as spiraling flows, boundary layer separation, and jet flows, which can be handled by the model [18].

An important advantage of the RSM is its ability to capture the anisotropic nature of turbulence, which is essential for flows with significant shear or strain. In addition, the model can capture the effects of pressure gradients on turbulence, which is crucial for predicting flow separation and reattachment. However, the RSM has certain restrictions. It is computationally expensive and requires substantial computing resources, which limits its use in large-scale simulations. In addition to requiring precise boundary conditions, the model can be sensitive to the selection of turbulence closure constants.

The RSM has been used in airfoil simulations to study the effects of turbulence on the aerodynamic performance of airfoils, including the lift and drag coefficients, the onset of flow separation, and the effects of turbulence on airfoil noise generation. Additionally, the RSM has been utilized to optimize the design of airfoils for specific applications, including wind turbines and aircraft.

The prediction of the advent of flow separation is one of the primary applications of the RSM in airfoil simulations. When the boundary layer on the airfoil surface becomes tumultuous and loses its ability to adhere to the surface, flow separation occurs. This causes a decrease in lift and an increase in drag, which can considerably impact the airfoil's performance. The RSM can accurately predict the location of the separation point and the resulting lift and drag coefficient changes [1].

The RSM has also been utilized to investigate the effects of turbulence on the noise generation of airfoils. Understanding turbulence behavior is essential for designing quieter airfoils, as turbulent flows are the primary source of disturbance in airfoil applications [8]. The RSM has been used to predict the levels of turbulence and noise generated by airfoils, as well as to optimize the design of airfoils for reduced noise levels.

In conclusion, the RSM is a higher-order turbulence model that can reliably predict the behavior of turbulent flows in simulations involving airfoil shapes. It can handle intricate flow conditions and capture the effects of pressure gradients on turbulent flow. The RSM is a potent instrument for optimizing airfoil designs for specific applications, such as wind turbines and aircraft, and predicting the onset of flow separation and airfoil noise generation, despite its limitations [2].

E. Large Eddy Simulation

Large Eddy Simulation (LES) is a CFD simulation used to predict turbulence in airfoil flows. It is a relatively new method that represents turbulent structures more precisely than the traditional RANS method. The LES approach has been extensively used in the study of complex turbulent flows such as those encountered in airfoil applications [2].

Separating the turbulent flow field into resolved and subgrid-scale components is the foundation of the LES method. The resolved components are the large-scale turbulent structures that can be resolved by the numerical grid, whereas the subgrid-scale components are the smaller-scale turbulent structures that cannot be resolved by the grid. Subgrid-scale components are modeled using a turbulence model, while resolved components are calculated explicitly.

LES provides a more accurate representation of turbulence than RANS because it resolves the larger-scale turbulent structures responsible for the majority of turbulent energy. The method is ideally adapted for high-Reynolds-number flows, such as those encountered in airfoil applications, in which larger-scale structures dominate the turbulent energy.

LES is able to capture irregular flow phenomena such as flow separation, vortex shedding, and turbulence transition, which is one of its advantages. In simulations of airfoils, the LES method has been applied to analyze the irregular flow behavior and predict the aerodynamic performance of airfoils under realistic flow conditions [7]. The method has been utilized to accurately predict the lift and drag coefficients, the pressure distribution on the airfoil surface, and the location and magnitude of flow separation [10].

The ability of the LES approach to accurately predict the noise generated by airfoils is another advantage. The turbulent flow over an airfoil generates noise, which is a significant factor in many airfoil applications, including aircraft and wind turbines. The LES method has been utilized to predict the noise emitted by airfoils and optimize airfoil designs for noise reduction [9].

The computational cost of the LES method is high because a narrow grid is required to resolve the larger-scale turbulent structures. However, advancements in computing technology have made the method more applicable to simulations of realistic airfoils. In airfoil simulations, the approach necessitates accurate boundary conditions and turbulence closure models, which can be a challenge [5].

In airfoil simulations, the LES method has been utilized to investigate a vast array of applications, including the optimization of airfoil designs for specific applications, such as wind turbines and aircraft. Airfoils' aerodynamic efficacy has also been studied in relation to flow conditions such as turbulence intensity and Reynolds number. The method has also been employed to investigate the effects of airfoil geometry on the unsteady flow behavior, including the formation and dispersal of vortices [7].

Predicting the onset of flow separation is one of the primary implementations of the LES method in airfoil simulations. Flow separation is a significant factor in the aerodynamic performance of airfoils, and is characterized by the separation of the boundary layer from the airfoil's surface. The LES method has been utilized to predict the location and magnitude of flow separation and to optimize airfoil designs for decreased flow separation.

In conclusion, the LES method is a potent instrument for predicting the irregular behavior of turbulent flows in applications involving airfoils. The approach provides a more accurate representation of the larger-scale turbulent structures and is well-suited to high-Reynolds-number flows. The method has been used to investigate a variety of applications in airfoil simulations, including the prediction of lift and drag coefficients, the effects of airfoil geometries on the unsteady flow behavior, and the prediction of noise emitted by airfoils. Airfoil designs for specific applications, such as wind turbines and aircraft, have also been optimized using this methodology. The LES method has helped researchers and engineers obtain a better understanding of the complex and unsteady behavior of turbulent flows in airfoil applications, and has contributed to the enhancement of the aerodynamic performance and efficiency of airfoils in a variety of applications [8].

IV. CONCLUSION

In conclusion, turbulence modeling is crucial for predicting the aerodynamic behavior of airfoils with precision. The choice of turbulence model depends on the application and the intended degree of precision. The SA model, k-epsilon model, k-omega model, RSM, and LES approach are the most frequently used turbulence models in airfoil simulations.

The SA model is a one-equation, simple, and computationally effective turbulence model. It has been extensively utilized in airfoil simulations and is suited to low Reynolds number flows. The k-epsilon model is a two-equation turbulence model that is both more complex and more accurate than the SA model. It is one of the most widely used turbulence models in CFD simulations and is suitable for a variety of airfoil applications. The k-omega model is a two-equation turbulence model analogous to the k-epsilon model, but it is more applicable to high Reynolds number flows. The RSM is a more complex turbulence model tailored to complex flows, such as those involving flow separation and recirculation. Lastly, the LES method is a computationally costly method that is well-suited to turbulent flows and has been applied to a wide variety of airfoil applications.

Each turbulence model has its own advantages and disadvantages, and the model chosen depends on the application and desired level of precision. The precision of the turbulence model is contingent upon the precision of the boundary conditions, grid resolution, and turbulence closure models. Therefore, it is essential to choose the turbulence model with care and validate the results whenever possible using experimental data.

Turbulence modeling is an indispensable instrument for the design and optimization of airfoils for a variety of applications, including wind turbines, aircraft, and hydrofoils. Predicting the aerodynamic performance of airfoils accurately can result in significant enhancements to efficiency, pollution reduction, and safety. As computational capacity and simulation techniques continue to advance, so will the accuracy and applicability of turbulence models in airfoil simulations, resulting in more efficient and effective airfoil designs.

REFERENCES

[1] A.C.K. Lai, W.W. Nazaroff, Modeling indoor particle deposition from turbulent flow onto smooth surfaces, J. Aerosol Sci. 31 (4) (2000) 463–476.

[2] B. Zhao, J. Wu, Modeling particle deposition from fully developed turbulent flow in ventilation duct, Atmos. Environ. 40 (3) (2006) 457–466.

[3] C. Chen, C.-H. Lin, D. Wei, Q. Chen, Modeling particle deposition on the surfaces around a multi-slot diffuser, Build. Environ. 107 (2016) 79–89.

[4] Q. Cao, C. Chen, S. Liu, C.-H. Lin, D. Wei, Q. Chen, Prediction of particle deposition around the cabin air supply nozzles of commercial airplanes using measured in-cabin particle emission rates, Indoor Air 28 (6) (2018) 852–865.

[5] Y. Pan, C.-H. Lin, D. Wei, C. Chen, Experimental measurements and large eddy simulation of particle deposition distribution around a multi-slot diffuser, Build. Environ. 150 (2019) 156–163.

[6] A.C.K. Lai, Particle deposition indoors: a review, Indoor Air 12 (4) (2002) 211–214.

[7] C. Chen, B. Zhao, C.J. Weschler, Indoor exposure to outdoor PM10: assessing its influence on the relationship between PM10 and short-term mortality in U.S. cities, Epidemiology 23 (6) (2012) 870–878.

[8] S. Shi, C. Chen, B. Zhao, Modifications of exposure to ambient particulate matter: tackling bias in using ambient concentration as surrogate with particle infiltration factor and ambient exposure factor, Environ. Pollut. 220 (2017) 337–347.

[9] X. Chen, A. Li, An experimental study on particle deposition above near-wall heat source, Build. Environ. 81 (2014) 139–149.

[10] X. Chen, A. Li, R. Gao, Numerical investigation on particle deposition in a chamber with an attached-wall heat source, Indoor Built Environ. 23 (5) (2013) 640–652.

[11] Y. Pan, C.-H. Lin, D. Wei, C. Chen, Experimental measurements and large eddy simulation of particle deposition distribution around a multi-slot diffuser, Build. Environ. 150 (2019) 156–163.

[12] E. Achenbach, Influence of surface roughness on the cross-flow around a circular cylinder, J. Fluid Mech. 46 (2) (1971) 321–335.

[13] Y. Wang, X. Fan, A. Li, L. Shang, H. Wang, Deposition of fine particles on vertical textile surfaces: a small-scale chamber study, Build. Environ. 135 (2018) 308–317.

[14] K.D. Squires, O. Simonin, LES–DPS of the effect of wall roughness on dispersed-phase transport in particle-laden turbulent channel flow, Int. J. Heat Fluid Flow 27 (4) (2006) 619–626.

[15] B. Zhao, J. Wu, Particle deposition in indoor environments: analysis of influencing factors, J. Hazard Mater. 147 (1) (2007) 439–448.

[16] Parsi, M., Vieira, R., Sajeev, S. K., McLaury, B. S., and S. A. Shirazi. "Experimental Study of Erosion in Vertical Slug/Churn Flow." Paper presented at the CORROSION 2015, Dallas, Texas, March 2015.

[17] Vieira, R. E., Sajeev, S., Shirazi, S. A., McLaury, B. S., & Kouba, G. (2015, June). Experiments and modelling of sand erosion in gas-liquid cylindrical cyclone separators under gas production and low-liquid loading conditions. In *17th International Conference on Multiphase Production Technology*. OnePetro.

[18] Sajeev, S. K. (2016). Sand Erosion of Gas-Liquid Cylindrical Cyclone Separators Under Gas Production and Low-Liquid Loading Conditions (Doctoral dissertation, University of Tulsa).

[19] Arabnejad, H., Sajeev, S., Guimmarra, A., Vieira, R., & Shirazi, S. A. (2016, September). Experimental Study and Modeling of Sand Erosion in the Gas-Liquid Cylindrical Cyclone GLCC Separators. In *SPE Annual Technical Conference and Exhibition*. OnePetro.

[20] Sajeev, S., McLaury, B., & Shirazi, S. (2017). Critical Velocities for Particle Transport from Experiments and CFD Simulations. *International Journal of Environmental and Ecological Engineering*, *11*(6), 548-552.

[21] Sajeev, S., McLaury, B. S., & Shirazi, S. A. (2018, June). Threshold particle concentration in single-phase and multiphase flow sand transport in pipelines. In *11th North American Conference on Multiphase Production Technology*. OnePetro.

[22] Sajeev, S. K. (2019). *Particle Transport in Horizontal Pipes for Single-Phase and Multiphase Flows at Very Low Concentrations Including the Threshold Concentration*. The University of Tulsa.

[23] Sajeev, S. K., McLaury, B. S., & Shirazi, S. A. (2019, June). Experiments and Modelling of Critical Transport Velocity of Threshold (Very Low) Particle Concentration in Single-Phase and Multiphase Flows. In *BHR 19th International Conference on Multiphase Production Technology*. OnePetro.

[24] Sajeev, S. (2023). 'An Overview of Project Management Application in Computational Fluid Dynamics'. World Academy of Science, Engineering and Technology, Open Science Index 195, International Journal of Industrial and Manufacturing Engineering, 17(3), 202 – 208.

Open Science Index, Aerospace and Mechanical Engineering Vol:17, No:10, 2023 publications.waset.org/10013261.pdf