International Journal of Engineering Applied Sciences and Technology, 2023
Vol. 8, Issue 01, ISSN No. 2455-2143, Pages 278-283
Published Online May 2023 in IJEAST (http://www.ijeast.com)

COMPARATIVE ANALYSIS OF TURBULENCE MODELS FOR SIMULATING FLOW OVER A FLAT PLATE.

Chintan R. Prajapati, Aniket P. Donga, Pranav R. Babariya and Akshar D. Patel
Department of Aeronautical Engineering,
Silver Oak College of Engineering and Technology
Ahmedabad, Gujarat, India - 382481

Abstract— **This research paper presents a comparative analysis of turbulence models for simulating the flow over a flat plate. The accurate prediction of turbulence characteristics is crucial for various engineering applications, including aerodynamics and heat transfer. In this study, three commonly used turbulence models, namely the k-ε, k-ω, and Laminar, are evaluated and compared for their ability to capture the flow behaviour over a flat plate. The simulations are conducted using Ansys, a widely-used computational fluid dynamics (CFD) software package. The flow conditions and geometry are carefully defined to ensure a consistent and controlled setup across the turbulence models. The key parameters of interest include boundary layer thickness, velocity profiles, and flow separation characteristics. The results obtained from each turbulence model are compared to provide insights into their performance and limitations. The comparative analysis focuses on the accuracy, computational efficiency, and ability to capture complex flow phenomena exhibited by the turbulence models. The findings of this study contribute to the understanding of the turbulence modelling techniques for simulating flow over a flat plate. The outcomes can aid researchers and engineers in selecting the most appropriate turbulence model for similar flow scenarios in various engineering applications.**

Keywords— **Turbulence models, flow simulation, flat plate, boundary layer, flow separation, skin friction coefficient, pressure distribution, wake region, computational fluid dynamics, numerical analysis.**

I. INTRODUCTION

The accurate simulation of fluid flow over a flat plate is of paramount importance in various engineering applications, including aerodynamics, heat transfer, and boundary layer analysis [1]. One critical aspect in achieving reliable simulations is the selection of an appropriate turbulence model [2]. Turbulence models play a crucial role in capturing the complex flow behaviour, especially near the surface of the flat plate [3]. They provide mathematical formulations to describe the turbulent eddies and their effects on the flow field.

The objective of this research paper is to perform a comprehensive comparative analysis of different turbulence models commonly used for simulating flow over a flat plate. The selected turbulence models will be evaluated based on their ability to accurately predict the key parameters of interest, such as boundary layer characteristics, skin friction, and separation points.

Choosing the right turbulence model holds paramount significance in capturing crucial physical phenomena within the flow, including the occurrence of boundary layer separation. It is imperative to select an appropriate turbulence model that can effectively represent these phenomena and provide accurate predictions [4]. Various turbulence models, including the Reynolds-Averaged Navier-Stokes (RANS) models, such as the standard k-ε, k-ω, as well as the Laminar model, will be investigated and compared in this study. Each model has its own assumptions and computational requirements, which can impact the accuracy and reliability of the simulation results. The appropriate selection of turbulence model can have major impact on the computational efficiency and the accuracy of the simulation [5].

By conducting a comparative analysis of these turbulence models, this research aims to provide valuable insights into their performance for simulating flow over a flat plate. The findings of this study will aid researchers and engineers in selecting the most appropriate turbulence model for their specific applications, ensuring accurate and efficient simulations.

In the following sections, we will discuss the methodology employed for the comparative analysis, provide a detailed description of the turbulence models under investigation, present the computational setup, and highlight the key parameters of interest. The results and discussions will shed light on the strengths and limitations of each turbulence model, culminating in valuable conclusions and recommendations for future studies in the field of fluid dynamics and aerodynamics.

International Journal of Engineering Applied Sciences and Technology, 2023
Vol. 8, Issue 01, ISSN No. 2455-2143, Pages 278-283
Published Online May 2023 in IJEAST (http://www.ijeast.com)

II. METHODOLOGY

A. Flow Configuration

First step in the simulation is to create flow configuration which is then simulated using ANSYS Fluent and created with the help of Design modular. Figure 1 shows the flow configuration of Flat plate.

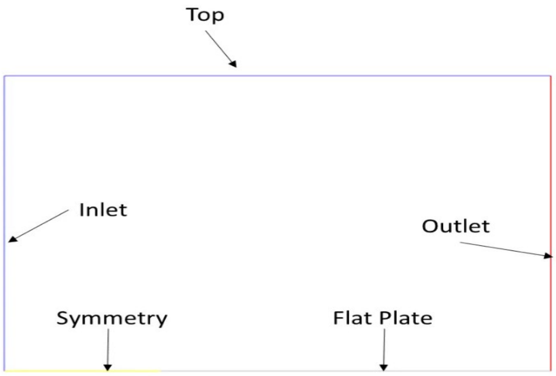

Fig. 1. Geometry of Fat plate

B. Geometry

The simulation was done using ANSYS Fluent. First of all, the problem statement was analysed and according that geometry was created. Figure 2 shows the basic geometry of the rectangle which was simulated. Here the five edges were considered namely, Inlet, Outlet, Flat plate, Top and Symmetry. The length of the plate was kept around 50 cm which is indicated in Figure 2 as H1, H3 in the Figure 2 depicts the symmetry with total length of 20 cm, and the height was 40 cm and shown as V2 in Figure 2. After completing sketch, surface was created by using tool called surface from sketch in ANSYS Fluent..

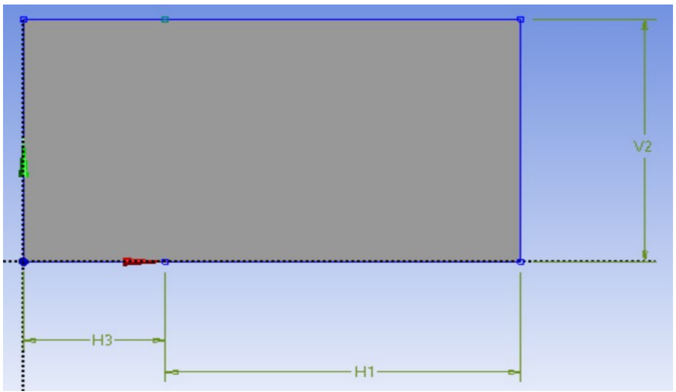

Fig. 2. Geometry of Fat plate

C. Mesh Generation

The meshing process begun with the generation of default mesh which is mostly unstructured which did not capture required condition for the simulation [6]. For our simulation, proper meshing should be ensured before the investigate the accuracy of the turbulence models used. To improve computational efficiency without sacrificing accuracy, the meshes [7]. we required mesh which is more refine above the flat plate to capture the flow field phenomenon i.e. turbulence, vortices, and flow patterns and to visualize the boundary layer over flat plate. In order to generate denser mesh near flat plate edge sizing was used and in that edge was divided into 100 divisions [8].

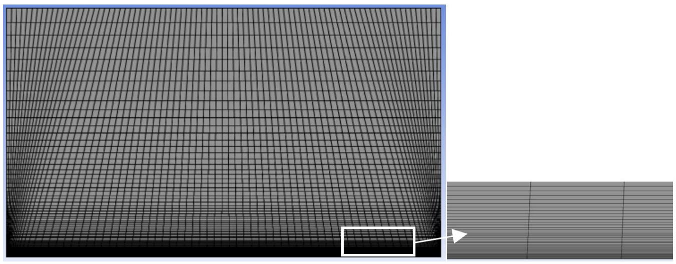

Fig. 3. Mesh

D. Physics Setup

In the third step, pressure based solver was used as flow over flat plate at lower velocities is incompressible hence this solver applied to the simulation [9]. Air was used as fluid over flat plate. The density and viscosity of the air was 1.225 Kg/m3, 1.7894×10^{-5} respectively. Boundary conditions are shown in Table 1.

Table -1 Boundary Conditions

Zone	Type	Value
Inlet	**Velocity-inlet**	**1 m/s**
Outlet	**Pressure-outlet**	**0 Pa[1]**
Top	**Wall**	**45.8103**
Flat plate	**No-slip wall**	**-**
Symmetry	**Symmetry**	**-**

[1] Relative to atmospheric condition

E. Solution

In the last step of simulation, where the Navier-stokes equation with the energy equation are solved [10]. Laminar viscous model was used with coupled scheme in the pressure-velocity coupling and second order upwind scheme was in the spatial discretization. Solution was monitored using a residual monitor with convergence criteria 10^{-6} [10].

III. RESULT

A. Laminar Model

Figure.4 shows the velocity distribution over the flat plate. The velocity is the lowest at the leading edge and increases as the length is increasing as well as velocity vector is shown in Figure.4 shows the development of boundary layer, the fluid particles are in the rest i.e. zero velocity due to the viscous effect.

Fig. 4. Velocity Contour (Laminar)

Figure.6 of pressure contour indicates that the pressure is highest at the leading edge and decreasing as we go to-ward outlet. Figure.4 & Figure.6 show that maximum velocity is 1.01587 m/s and maximum pressure is 0.048959 Pa.

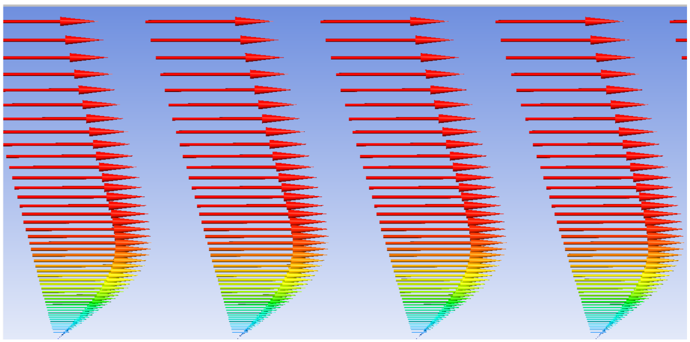

Fig. 5. Velocity vector

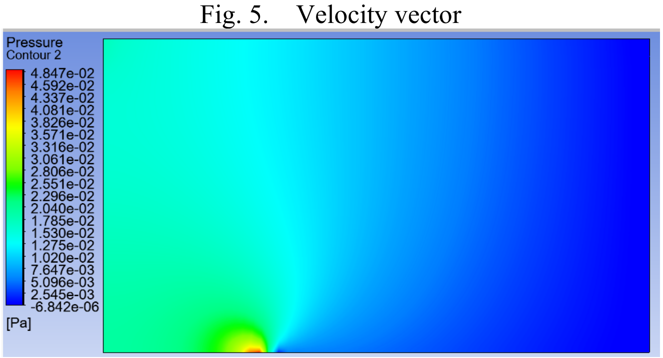

Fig. 6. Pressure Conture (Laminar)

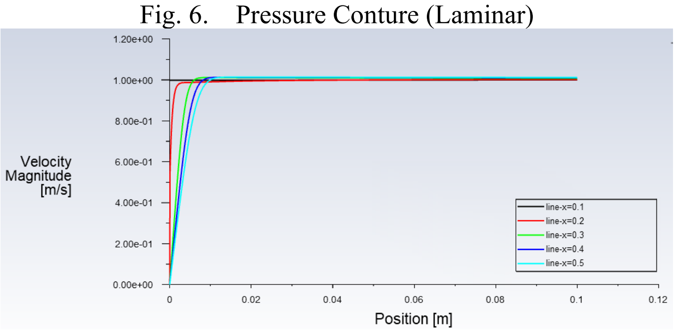

Fig. 7. Velocity magnitude at different location

Figure 7 shows the velocity is increasing as we move toward outlet; however after some distance velocity remains constant as shown in figure 7.

B. **K-Ɛ Model**

In the k-epsilon model upstream conditions were the same as shown in Table 1. Energy equation is solved and Laminar viscous model was and second order upwind scheme was in the spatial discretization for turbulent kinetic energy and dissipation rate [11].

Fig. 8. Velocity Contour (k-epsilon)

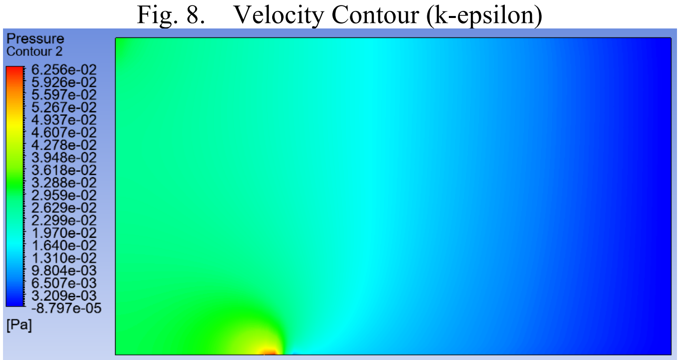

Fig. 9. Pressure Contour (k-epsilon)

As the figure 8 and 9 suggest that velocity and pressure contour are almost same, but main differen is in the velocity and pressure which is increased slightly to 1.02373 m/s and 0.0630674 Pa respectively. The trend of increase and decrase is same in the velocity and pressure.

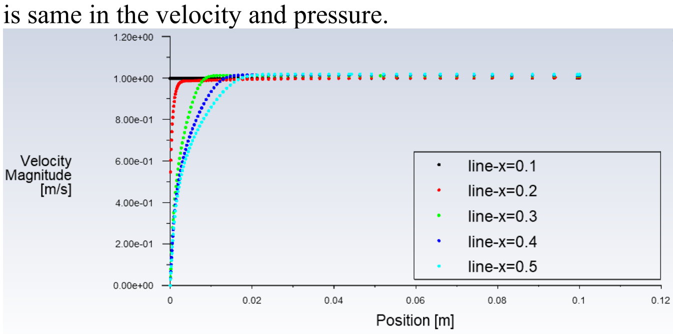

Fig. 10. Velocity magnitude at different location

Figure 10 shows the velocity profile at different location, it is clear that velocity at line-x=0.1 is remain same but at the different location velocity is increasing exponentially which can be seen from the figure.

C. K- ω Model

As we are comparing different turbulence model so all the procedure is same i.e. all boundary conditions.

Fig. 11. Velocity Contour (k-omega)

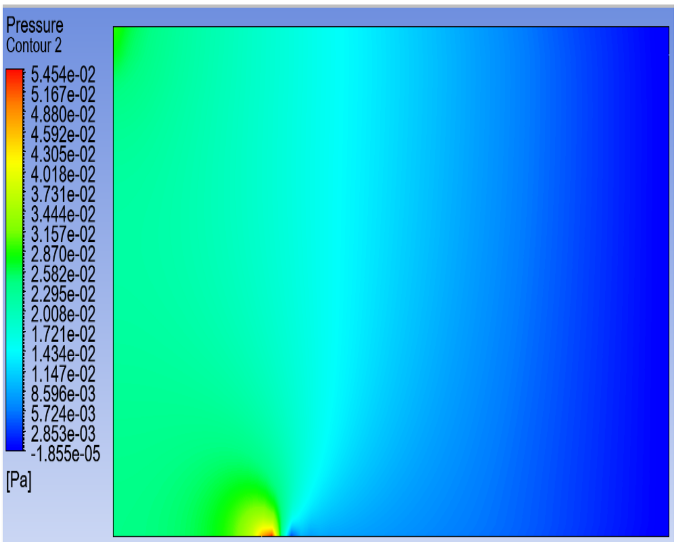

Fig. 12. Pressure Contour (k-epsilon)

Velocity is almost same in the free stream condition but on the flat plate due to viscous force velocity is almost 0 near the wall [12]. From Figure 11 it is clear that maximum velocity is 1.01925 m/s and Figure 12 shows the maximum pressure is 0.0549798 Pa. Figure 13 is almost identical to Figure 7 which is velocity profile of laminar model.

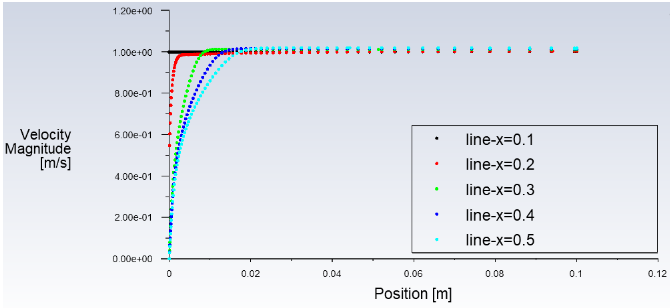

Fig. 13. Velocity magnitude at different location

Comparison of skin friction of different model is compared in the figure 14 and it indicates that laminar model and k-omega model exhibits same skin friction coeffcient at the leading edge. The skin friction coefficint is 0.0506003 Pa at the leading edge.

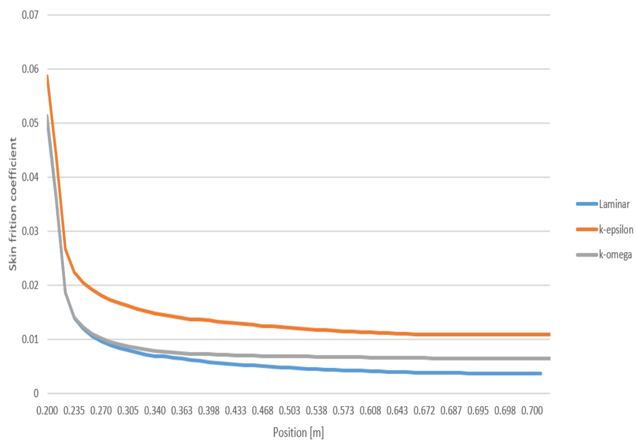

Fig. 14. Velocity Contour (k-omega)

Table -2 Comparison of different Turbulence model

Turbulence model	**Max. Velocity**	**Max. Pressure**
Laminar	**1.01587 m/s**	**0.048959 Pa**
k-epsilon	**1.02373 m/s**	**0.0630674 Pa**
k-omega	**1.01925 m/s**	**0.0549798 Pa**

IV. EFFECT OF CHANGING THE LENGHT OF THE PLATE

The velocity distribution and boundary layer thickness are influenced by the length of the plate. As depicted in Figure 15, increasing the horizontal dimension of the plate leads to a decrease in velocity and an increase in boundary layer thickness. Specifically, the maximum velocity occurs at a plate length of 0.8, while longer lengths result in reduced velocity and thicker boundary layers [10] [13].

International Journal of Engineering Applied Sciences and Technology, 2023
Vol. 8, Issue 01, ISSN No. 2455-2143, Pages 278-283
Published Online May 2023 in IJEAST (http://www.ijeast.com)

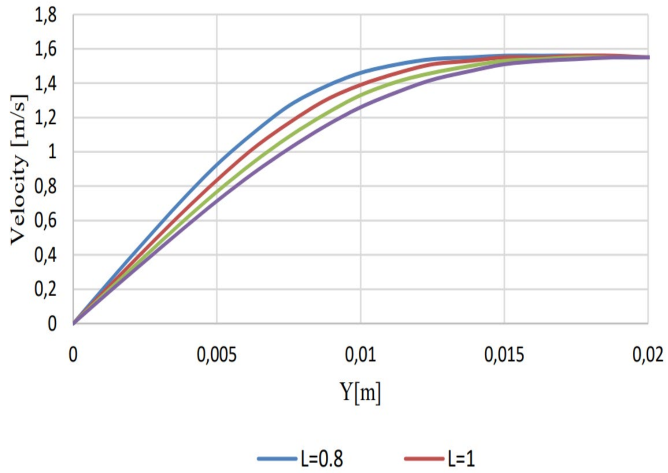

Fig. 15. Velocity distribution and boundary layers for different plate's length [10]

V. CONCLUSION

In conclusion, this research paper presented a comparative analysis of turbulence models for simulating the flow over a flat plate. The study evaluated three commonly used turbulence models, namely the k-ε, k-ω, and Laminar models, in terms of their ability to capture the flow behaviour accurately. The simulations were performed using Ansys software, and the results were compared.

Based on the analysis, several key findings emerged. Firstly, the selection of the turbulence model significantly influenced the accuracy and computational efficiency of the simulations. The k-ε and k-ω models exhibited good performance in capturing the boundary layer thickness, velocity profiles, and flow separation characteristics. However, the Laminar model, despite being a simplified model, showed limitations in accurately predicting certain flow phenomena.

The comparative analysis provided insights into the strengths and limitations of each turbulence model. The k-ε and k-ω models demonstrated their capability to capture complex flow behaviour near the flat plate, making them suitable for various engineering applications. However, it is important to consider the computational cost associated with these models, as they require more computational resources compared to the Laminar model.

This research contributes to the understanding of turbulence modelling techniques for flow over a flat plate. The findings can guide researchers and engineers in selecting the most appropriate turbulence model for similar flow scenarios in various engineering applications. Furthermore, the study highlights the importance of validating simulation results against experimental data and benchmark solutions to ensure the accuracy and reliability of the chosen turbulence model.

In future research, it would be beneficial to explore other turbulence models and investigate their performance in simulating flow over a flat plate. Additionally, conducting experimental studies to validate the simulation results further would enhance the credibility of the findings. Overall, this research serves as a foundation for further advancements in the field of fluid dynamics and aerodynamics, aiding in the development of more accurate and efficient simulation techniques.

Acknowledging the limitations of this study, it is recommended to consider more complex geometries and flow conditions in future investigations to broaden the understanding of turbulence models in practical engineering scenarios. By addressing these aspects, researchers can continue to refine and improve turbulence modelling techniques, contributing to advancements in various engineering fields.

In conclusion, this research provides valuable insights into the comparative analysis of turbulence models for simulating flow over a flat plate. The findings contribute to the existing knowledge and can guide future research and applications in the field of fluid dynamics and aerodynamics.

VI. REFERENCE

[1] Taha, H. E., & Gonzalez, C. (2023). Refining Kutta's flow over a flat plate: Necessary conditions for lift. AIAA Journal, 61(5), 2060–2068. doi:10.2514/1.j062273

[2] Sepka, S., & Tauber, M. (2009). Turbulent flow over an ablating flat plate with roughness. 41st AIAA Thermophysics Conference. doi:10.2514/6.2009-4079

[3] Rubesin, M. W., & Johnson, H. A. (1949). A critical review of skin-friction and heat-transfer solutions of the laminar boundary layer of a flat plate. Journal of Fluids Engineering, 71(4), 383–388. doi:10.1115/1.4017082

[4] Araya, G. (2019). Turbulence model assessment in compressible flows around complex geometries with unstructured grids. Fluids, 4(2), 81. doi:10.3390/fluids4020081

[5] Gooding, W. J., Meier, M. A., & Key, N. L. (2021). The impact of various modeling decisions on flow field predictions in a centrifugal compressor. Journal of Turbomachinery, 143(10). doi:10.1115/1.4050674

[6] Yokoi, Y. (2019). Numerical experiment of flow around a flat plate with round edges. 2019 IEEE 10th International Conference on Mechanical and Aerospace Engineering (ICMAE). doi:10.1109/icmae.2019.8880983

[7] Igali, D., Mukhmetov, O., Zhao, Y., Fok, S. C., & Teh, S. L. (2019). Comparative analysis of turbulence models for automotive aerodynamic simulation and Design. International Journal of Automotive Technology, 20(6), 1145–1152. doi:10.1007/s12239-019-0107-7

[8] Yokoi, Y. (2020). Numerical experiment of flow around a compound airfoil which consists of two round flat plates. 2020 11th International Conference on Mechanical and Aerospace Engineering (ICMAE). doi:10.1109/icmae50897.2020.9178883

[9] Shahmohamadi, H., & Rashidi, M. M. (2017). Experimental investigation and a novel analytical solution of turbulent boundary layer flow over a flat plate in a wind tunnel. International Journal of Mechanical Sciences, 133, 121–128. doi:10.1016/j.ijmecsci.2017.08.043

[10] Klazly, M. M., & Bognár, G. (2019). Computational fluid dynamic simulation of laminar flow over a flat plate. Design of Machines and Structures, 9(1), 29–47. doi:10.32972/dms.2019.004

[11] Kaffash, M. H., Ganji, D. D., & Nobakhti, M. H. (2017). An analytical solution of turbulent boundary layer fluid flow over a flat plate at high Reynolds number. Journal of Molecular Liquids, 230, 625–633. doi:10.1016/j.molliq.2017.01.009

[12] Sarker, K., Ali, M., & Islam, Q. (2014). A numerical study on the physics of flow over a flat plate with backward facing step. Procedia Engineering, 90, 351–357. doi:10.1016/j.proeng.2014.11.861

[13] Canbolat, G., Yıldızeli, A., Köse, H. A., & Çadırcı, S. (2018). Numerical investigation of transitional flow over a flat plate under constant heat fluxes. Academic Perspective Procedia, 1(1), 187–195. doi:10.33793/acperpro.01.01.39