Advanced and Sustainable Technologies
ASET
E-ISSN 2976-2294

Volume 3, No 2, December 2024 [71-79]

Influence of Inlet Mass Flow Rates on Fluid Characteristics of Engine Exhaust Manifold

Vinoth Baskaran[1*]

[1]Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis (UniMAP), 02600 Arau, Perlis, Malaysia.

Received 19 August 2024, Revised 10 September 2024, Accepted 17 September 2024

ABSTRACT

This study presents a comprehensive Computational Fluid Dynamics (CFD) analysis of internal flow behavior within an engine exhaust manifold under various operating conditions. The primary focus is on examining velocity and pressure distributions across different scenarios. The simulations explore the effects of varying inlet mass flow rates (0.004 m^3/s, 0.006 m^3/s, and 0.008 m^3/s) using three turbulence models: k-ε, k-ω, and k-ω SST. The analysis provides detailed insights into complex flow patterns, including meshing strategies, boundary condition setups, and solver configurations, with particular attention to phenomena such as flow separation, recirculation, and turbulence. Parametric variations are thoroughly examined to assess how different turbulence models and mass flow rates influence critical parameters like pressure and velocity. The findings offer valuable insights into the manifold's flow dynamics, contributing to design enhancements to improve engine performance and reduce emissions.

Keywords: Engine exhaust manifold, computational fluid dynamics, Simflow 4.0, internal flow, Turbulent model.

1. INTRODUCTION

The efficient design and performance optimization of engine exhaust manifolds is critical for enhancing overall engine efficiency, boosting power output, and reducing emissions [1]. The exhaust manifold serves as a conduit that collects exhaust gases from multiple cylinders and channels them into a single pipe, directing these gases toward the exhaust system. The internal flow dynamics within the manifold, characterized by complex phenomena [2], such as flow separation, recirculation, and turbulence, play a pivotal role in determining the engine's performance and emission characteristics. These complex flow behaviors can lead to uneven exhaust gas distribution, pressure losses, and hotspots, adversely affecting engine efficiency and increasing emissions [3].

Given the complexity of these flow patterns, a comprehensive understanding is essential for optimizing manifold design [4]. This is where Computational Fluid Dynamics (CFD) proves invaluable. CFD has become a powerful tool in analyzing and optimizing exhaust manifold flow characteristics [5], allowing engineers to simulate and examine the behavior of gases within the manifold under various operating conditions. By providing detailed insights into velocity and pressure distributions, CFD enables the identification of potential issues [6] and the implementation of design improvements [7] that can enhance engine performance, lower fuel consumption, and meet stringent emission standards. This approach leads to more efficient manifold designs and contributes to developing cleaner and more powerful engines.

*Corresponding author: vinothbaskaran5611@gmail.com

Several studies have explored the application of CFD in analyzing the internal flow of engine exhaust manifolds. Sunny Manohar & Krishnaraj [8] conducted a CFD analysis on a four-cylinder engine exhaust manifold, highlighting the importance of optimizing the geometry to reduce pressure drop and enhance flow uniformity. Their study demonstrated that strategic design modifications could significantly improve exhaust gas flow, boosting engine efficiency. Similarly, Nouhaila et al. [9] examined the impact of a crack in the exhaust manifold system of an internal combustion engine, focusing on how it affects pressure drop and flow patterns. The presence of a crack increases the pressure drop, which diminishes engine efficiency and alters exhaust gas flow by introducing turbulence and vortex formation. These changes reduce performance and compromise the manifold's structural integrity by raising the coefficient of friction. Recent advancements in CFD technology have enabled more detailed and accurate simulations of complex flow phenomena. Benek and Ozsoysal [10] utilized the k-ω SST model to study the effects of varying inlet mass flow rates on the performance of a turbocharged engine exhaust manifold. Their findings indicated that higher mass flow rates increased turbulence and pressure drop, which could adversely affect the engine's efficiency. Besides, they emphasized the role of meshing techniques and solver configurations in achieving reliable simulation results. Their studies collectively highlighted the critical factors influencing exhaust manifold performance and the potential of CFD to provide valuable insights for design optimization.

Sadhasivam et al. [11] used CFD analysis to predict and mitigate temperature, speed, and pressure distribution in exhaust manifolds prone to cracking damage due to high operational temperatures. The study provided insights into potential material stress and deposit formation by analyzing fluid flow, temperature, and stress. They recommended the design improvements aimed at reducing temperature peaks and gradients, thereby extending the service life of the exhaust manifold and enhancing overall engine performance. Desai et al. [12] conducted a CFD analysis to evaluate the impact of velocity, temperature, and back pressure on the volumetric efficiency of exhaust manifolds, comparing two different designs. They also examined the effects of various fuels, including LPG, alcohol, and gasoline, on overall efficiency. Their results indicated that the type A manifold, which exhibited higher pressure values, outperforms type B in terms of performance and efficiency, particularly with gasoline fuel, which showed lower pressure and velocity. Venkatesan et al. [13] reported that the design geometry of an exhaust manifold is crucial for ensuring smooth combustion and reducing emissions in spark ignition engines. They analyzed and compared the exhaust gas back pressures and velocities across different manifold models under various engine load conditions to identify the optimal design. They revealed that model 5 is the most effective at controlling emissions, making it the recommended choice for use in multi-cylinder engines. The analysis was conducted using virtual models, with modeling and simulations performed using CATIA v5 and ANSYS software. With increasing air pollution regulations, exhaust systems have become critical components, requiring complex design processes that balance emissions and power consumption. Bral et al. [14] provided the necessary dimensions and parameters to model an exhaust system for a typical compact goods truck. Computational Fluid Dynamics (CFD) analysis compared three operating conditions, ultimately identifying the most effective exhaust manifold design. Nouhaila et al. [15] evaluated the accuracy of various turbulence models in simulating exhaust manifold flow using CFD, aiming to identify high-stress areas, minimize pressure drop, and optimize exhaust gas flow. By comparing five commonly used turbulence models, the research provides valuable insights into their effectiveness in predicting flow characteristics, both quantitatively and qualitatively. The k-kl-ω model emerged as the most effective in capturing transitional flow effects. In contrast, the k-ω STD and SST transition models demonstrated higher turbulent kinetic energy (TKE), indicating better performance in modeling complex flows. Despite variations in turbulence predictions, all models showed similar pressure drop trends, offering critical guidance for optimizing exhaust manifold design to enhance engine performance, reduce emissions, and improve durability.

This project presents a comprehensive CFD analysis of the internal flow within an engine exhaust manifold with four inlets and one outlet under varying conditions. The primary objectives are to study the velocity and pressure distributions within the manifold. The simulation investigated two independent parameters: inlet mass flow rates (0.004 m^3/s, 0.006 m^3/s, and 0.008 m^3/s) and three different turbulence models (k-ε, k-ω, and k-ω SST). The analysis involves thorough meshing, boundary condition setup, and solver configuration to capture complex flow patterns accurately. Steady-state simulations are performed to provide a detailed understanding of the flow dynamics. By analyzing the parametric variations, the study aims to determine how the type of turbulent flow and the mass flow rate affect critical parameters like pressure and velocity. The results are expected to offer valuable insights into the manifold's flow dynamics, informing design improvements for enhanced engine performance and reduced emissions.

2. MATERIAL AND METHODS

Computational Fluid Dynamics (CFD) is performed to study the flow characteristics, such as velocity and pressure distributions in the exhaust manifold. This technology has numerous applications in the automotive industry, aiding in designing and optimizing the manifold design, controlling emissions, and reducing development costs. The geometry of the engine exhaust manifold is shown in Figure 1. Using the SimFlow 4.0 software [16], the geometry of the exhaust manifold was imported for meshing and simulation setting to estimate the maximum velocity and pressure when the inlet velocity or mass flow rate varies from 0.004 m^3/s, 0.006 m^3/s, and 0.008 m^3/s using the RANS k-ε, RANS k-ω, and RANS k-ω SST turbulent models. The base mesh parameters were set to auto-size at maximum (m) (0.44691, 0.1913, 0.21934) to create the domain of the flow. The mesh division of (50, 50, 50) was set on the domain to generate the hexahedral dominant meshing on the domain and model. The cell size at the x-, y- and z-axis is 0.010076, 0.0052338, and 0.0044086m, respectively. The total generated mesh elements are 13760. The boundary conditions, such as inlet velocity, pressure outlet, and walls, were defined on the domain surfaces. In this simulation, the steady state, incompressible flow, and SIMPLE [17] discretization solver were considered based on the assumptions.

Figure 1 illustrates the geometry and boundary conditions of the engine exhaust manifold. This provides a foundation for understanding the computational domain used in the simulations. The geometry details the physical structure of the manifold, which impacts how the exhaust gases flow through it. The boundary conditions define the limits within which the simulations are conducted, such as inlet and outlet conditions, pressure, and temperature. Figure 2 shows the meshing process, a critical step in preparing the computational model for analysis. Figure 2 (a) shows the box-meshing method, which involves dividing the fluid domain into smaller and manageable elements that can be analyzed computationally. Figure 2 (b) presents the meshed model of the fluid domain. The mesh's quality and resolution significantly affect the simulation results' accuracy, with finer meshes generally leading to more precise outcomes but requiring more computational resources.

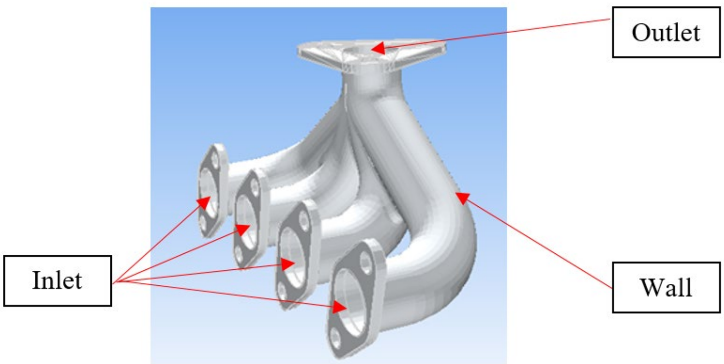
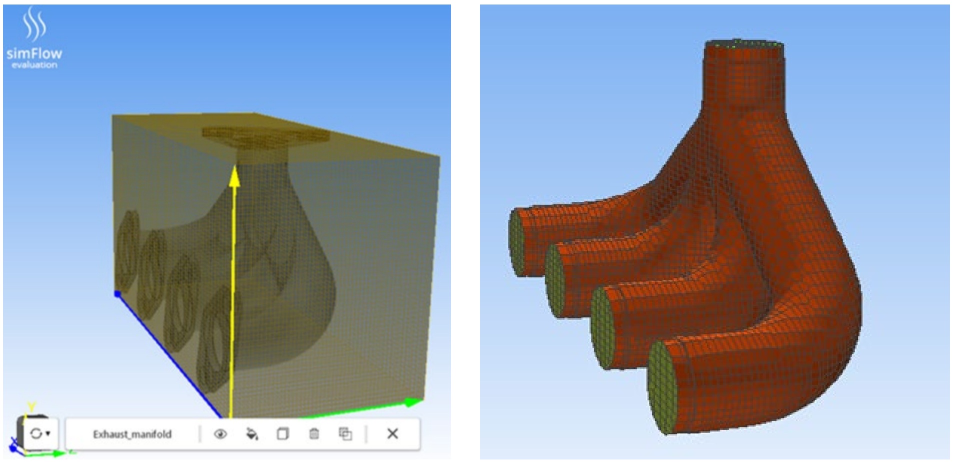


Figure 1: Geometry and boundary conditions of the engine exhaust manifold.



(a) (b)

Figure 2: (a) Box-meshing method and (b) meshed model of the fluid domain.

Figure 3 compares the residual plots for various mass flow rates using k-ω and k-ω SST turbulence models. Residual plots are used to assess the convergence of the numerical solution during simulation. Lower residual values indicate better convergence and more reliable results. The figure shows that the residuals decrease as the simulation progresses, with comparisons between different turbulence models. These plots help validate the choice of turbulence model and ensure that the simulations are robust and accurate, where all residuals met the convergence criteria below 1×10^{-4}.

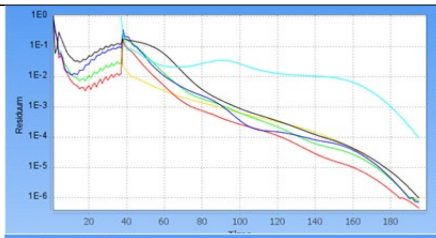
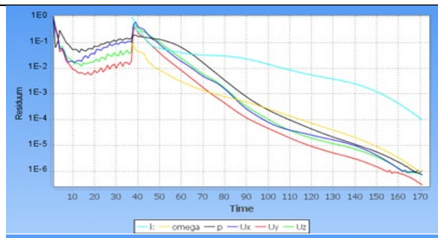
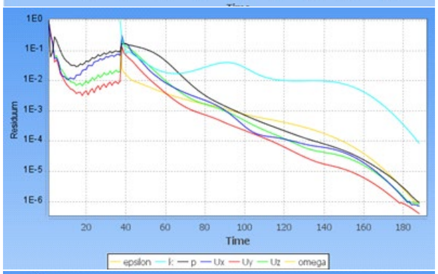
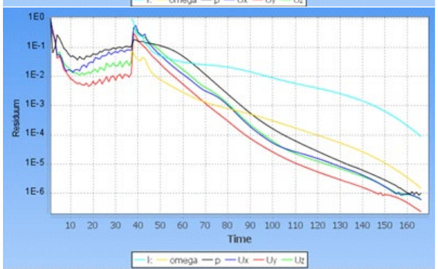
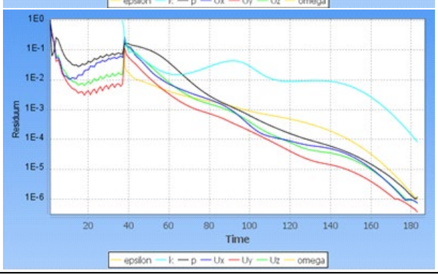
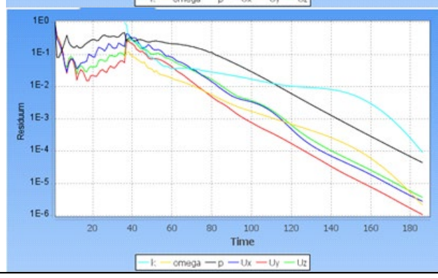
Mass Flow Rate (m^3/s)	k-ω	k-ω SST
0.004		
0.006		
0.008		

Figure 3: Residual plots for different mass flow rates with turbulence models.

3. RESULTS AND DISCUSSION

The maximum velocity increases with the mass flow rate for both turbulence models. This is expected as a higher mass flow rate signifies more exhaust gas flowing through the manifold per unit of time, which leads to higher velocities [18]. Table 1 summarizes the maximum velocity achieved in the exhaust manifold for different mass flow rates and turbulence models. The comparison of maximum velocities across the k-ε, k-ω, and k-ω SST models highlights the sensitivity of the flow field to the choice of turbulence model. Notably, the velocities increase with higher mass flow rates, and the differences between models may indicate varying levels of accuracy or applicability for different flow regimes. The k-ε model predicts slightly higher maximum velocity than other turbulence models because it tends to overestimate the energy associated with large-scale turbulent eddies. This model assumes that turbulence is isotropic and primarily governed by two transport equations: one for the turbulent kinetic energy (k) and another for the dissipation rate of this energy (ε). Due to its formulation, the k-ε model can more aggressively predict kinetic energy transfer from turbulence into mean flow, leading to higher velocity estimates, especially in regions of high shear or flow separation. This behavior is particularly evident at varying mass flow rates, where the model may not fully capture the effects of smaller eddies or transitional flow, resulting in a bias toward higher velocity predictions.

Table 2 details the maximum pressure within the exhaust manifold for different mass flow rates and turbulence models. Pressure is a critical parameter in assessing the performance and safety of the exhaust system. The variations observed among the models, particularly at lower mass flow rates, suggest that some models may overpredict or underpredict the pressure, affecting the design and analysis of the manifold. Figure 4 depicts the maximum velocity in the manifold for various mass flow rates using different turbulence models. This figure directly compares the velocity trends identified in Table 1. Figure 4 illustrates the relationship between maximum velocity in the manifold and mass flow rate for different turbulence models. The results indicate a strong linear correlation between the two variables for all models, as evidenced by the high R^2 values (0.9952, 1, and 0.9959). The k-ω model consistently predicts the highest maximum velocity across all mass flow rates, followed by the k-ω SST model. The k-ε model, on the other hand, consistently predicts the lowest maximum velocity. Besides, Figure 5 illustrates the maximum pressure data presented in Table 2. The results indicate a non-linear correlation between the two variables, as evidenced by the quadratic nature of the fitted equations for all models. The k-ε model predicts the highest maximum pressure across all mass flow rates, followed by the k-ω SST model. The k-ω model, on the other hand, predicts the lowest maximum pressure. Overall, the choice of turbulence model significantly impacts the predicted maximum velocity and pressure in the manifold, especially at higher mass flow rates.

Table 1: Maximum velocity for different mass flow rates with turbulent models.

Mass Flow Rate (m^3/s)	Maximum Velocity (m/s)		
	(k-ε)	(k-ω)	(k-ω SST)
0.004	1.5930	1.4811	1.4670
0.006	2.3885	2.2268	2.2270
0.008	3.1772	3.1772	3.1774

Table 2: Maximum pressure for different mass flow rates with turbulent models.

Mass Flow Rate (m^3/s)	Maximum Pressure (Pa)		
	(k-ε)	(k-ω)	(k-ω SST)
0.004	1.2995	17.5140	3.5020
0.006	1.4292	6.8811	6.0496
0.008	5.1302	5.1271	5.1300

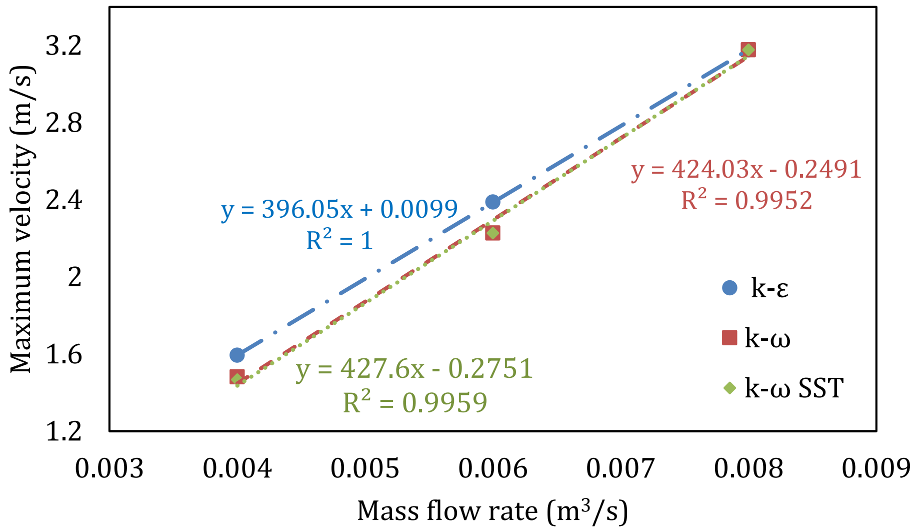


Figure 4: Maximum velocity in the manifold for different mass flow rates with turbulence models.

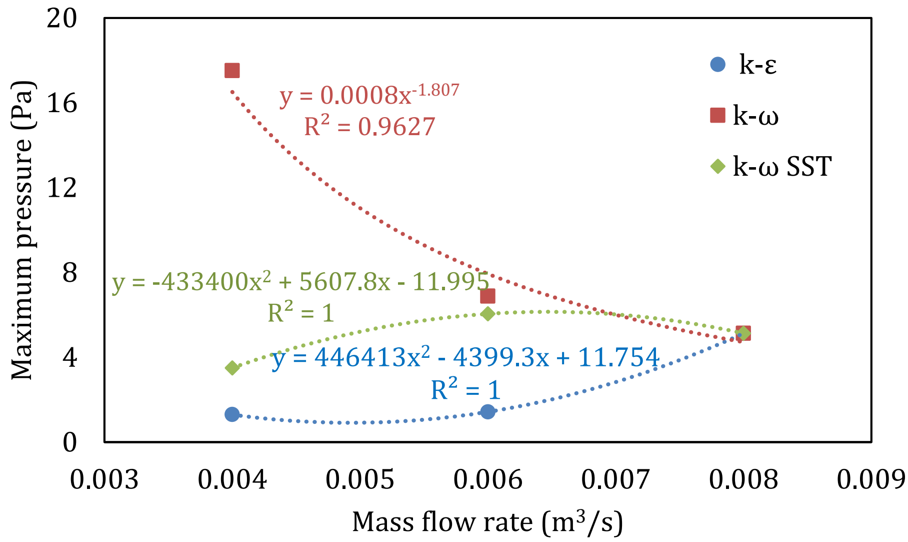


Figure 5: Maximum pressure in the manifold for different mass flow rates with turbulence models.

Figure 6 compares the velocity contours within the manifold at different mass flow rates using various turbulence models. Contour plots provide a detailed view of the flow distribution throughout the manifold, showing high and low-velocity regions. These plots are essential for identifying potential issues such as flow separation, recirculation zones, or areas of high shear, which could impact the performance and durability of the manifold. The k-ε model generally predicts higher velocities and more intense turbulence, while the k-ω model predicts lower and less intense turbulence. The k-ω SST model offers a balance between the two.

Figure 7 shows the pressure contours at different mass flow rates using various turbulence models. Like velocity contours, pressure contours help understand the pressure distribution within the manifold. These plots reveal high-pressure areas that may be prone to failure or require reinforcement. The comparison across turbulence models helps select the most accurate model for predicting pressure distribution under different operational conditions. As the mass flow rate increases, the overall pressure level within the manifold rises due to increased fluid momentum. The choice of turbulence model significantly affects the pressure distribution. The k-ε model generally predicts higher pressures, especially near the inlet and outlet sections, suggesting more intense turbulence and greater energy dissipation. The k-ω model, on the other hand, predicts lower pressures, indicating less turbulence and a more uniform pressure distribution. The k-ω SST model lies between the k-ε and k-ω models, balancing pressure prediction and turbulence intensity.

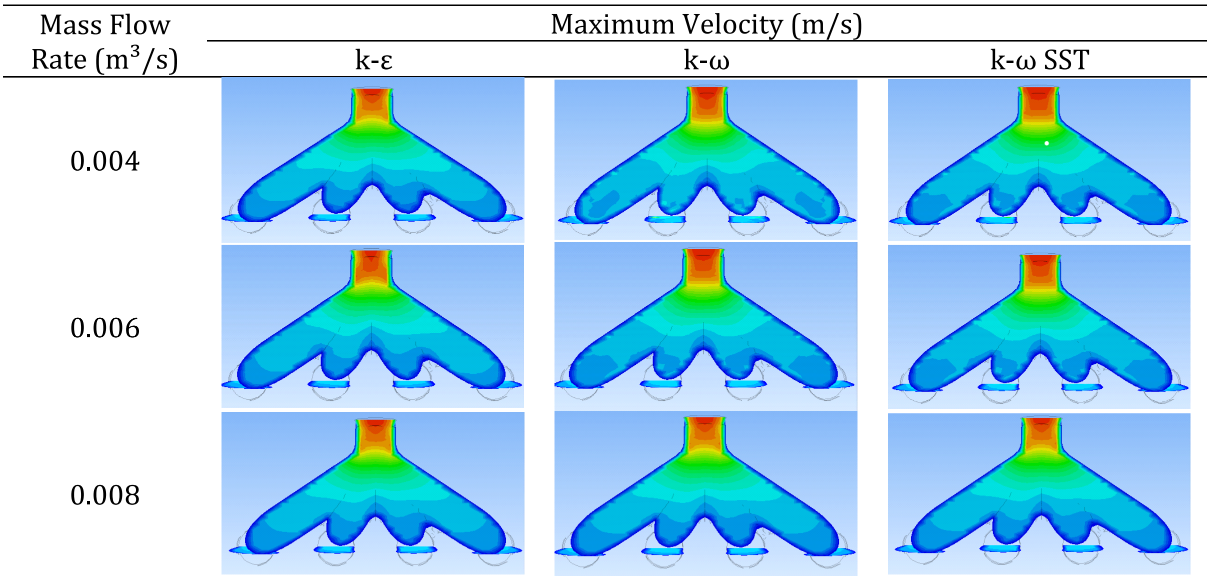


Figure 6: Comparison of velocity contour at different mass flow rates with turbulence models.

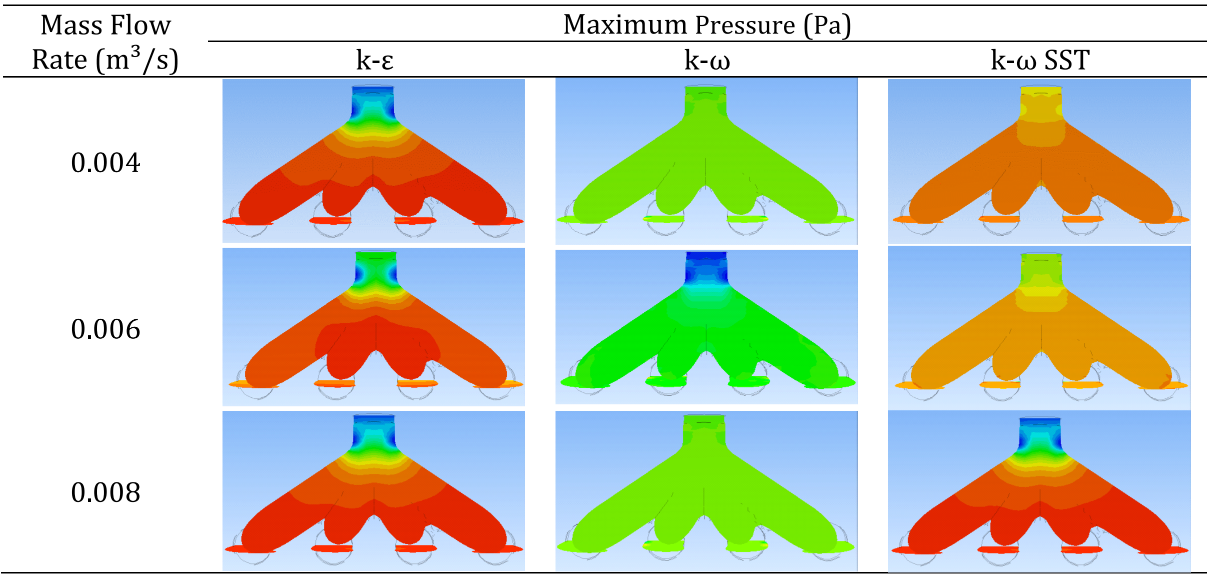


Figure 7: Comparison of pressure contour at different mass flow rates with turbulence models.

4. CONCLUSION

This CFD analysis assessed the internal flow characteristics of an engine exhaust manifold using two turbulence models: k-ε, k-ω and k-ω SST. The study focused on pressure and velocity distributions at various mass flow rates (0.004 m^3/s, 0.006 m^3/s, 0.008 m^3/s). The key findings include that the k-ω SST model consistently predicted higher maximum velocities than the k-ω model across all flow rates. Besides, the k-ω model displayed a counterintuitive decrease in maximum pressure with increasing flow rate. In contrast, the k-ω SST model exhibited a more expected pressure increase. This discrepancy suggests that the k-ω model may struggle with accurately capturing pressure behavior in complex geometries and strong pressure gradients. The results emphasize the importance of choosing a suitable turbulence model for CFD simulations of exhaust manifolds. The k-ω SST model appears more reliable for predicting pressure behavior in such complex systems. The unexpected results from the k-ω model highlight the need for careful verification and validation of CFD simulations, mainly when using simpler turbulence models. Future work should involve refining the mesh, improving convergence, and exploring alternative turbulence models, including SST variants. Additionally, validating simulations with experimental data, if available, and investigating the effects of different manifold designs on flow characteristics and pressure distribution could further enhance the accuracy and applicability of the analysis.

ACKNOWLEDGEMENTS

The author sincerely appreciates the Faculty of Mechanical Engineering & Technology's crucial support and resources during this research. Thanks to the CFD course coordinator for his professional guidance and mentorship.

REFERENCES

[1] Doppalapudi, A. T., Azad, A. K., & Khan, M. M. K. Combustion chamber modifications to improve diesel engine performance and reduce emissions: A review. Renewable and Sustainable Energy Reviews, vol 152, (2021) p. 111683.

[2] Onorati, A., Ferrari, G., & D'Errico, G. Fluid dynamic modeling of the gas flow with chemical specie transport through the exhaust manifold of a four cylinder SI engine. SAE transactions, (1999) pp. 819-834.

[3] Guan, B., Zhan, R., Lin, H., & Huang, Z. Review of the state-of-the-art of exhaust particulate filter technology in internal combustion engines. Journal of environmental management, vol 154, (2015) pp. 225-258.

[4] Teja, M. A., Ayyappa, K., Katam, S., & Anusha, P. Analysis of exhaust manifold using computational fluid dynamics. Fluid Mech Open Acc, vol 3, issue 1 (2016) p.1000129.

[5] Zhang, X., & Romzek, M. Computational fluid dynamics (CFD) applications in vehicle exhaust system, SAE Technical Paper, (2008), No. 2008-01-0612.

[6] Teja, M. A., Ayyappa, K., Katam, S., & Anusha, P. Analysis of exhaust manifold using computational fluid dynamics. Fluid Mech Open Acc, vol 3, issue 1 (2016) p.1000129.

[7] Hopf, A., Bartsch, G., Krämer, F., & Weber, C. CFD topology and shape optimization for port development of integrated exhaust manifolds, SAE Technical Paper, (2017), No. 2017-01-1339.

[8] Manohar, D. S., & Krishnaraj, J. Modeling and analysis of exhaust manifold using CFD. In IOP Conference Series: Materials Science and Engineering, IOP Publishing, vol 455, issue 1 (2018) p. 012132).

[9] Nouhaila, O., & Hassane, M. Analyzing the Impact of Cracks on Exhaust Manifold Performance: A Computational Fluid Dynamics Study. International Journal of Heat & Technology, vol 42, issue 2 (2024).

[10] Benek, G., & Ozsoysal, O. A. Influences of the dead end on the flow characteristics at the exhaust manifold of a marine diesel engine. Journal of Thermal Engineering, vol 7, issue 6 (2021) pp. 1519-1530.
[11] Sadhasivam, C., Murugan, S., Vairamuthu, J., & Priyadharshini, S. M. Design and analysis of two-cylinder exhaust manifold with improved performance in CFD. Materials Today: Proceedings, vol 37, (2021) pp. 2141-2144.
[12] Desai, A. R., Buradi, A., Gowthami, L., Praveena, B. A., Madhusudhan, A., & Bora, B. J. (2022, October). Computational Investigation of Engine Exhaust Manifold with Different Alternative Fuels By Using CFD. In 2022 IEEE 2nd Mysore Sub Section International Conference (MysuruCon), IEEE, (2022) pp. 1-6.
[13] Venkatesan, S. P., Ganesan, S., Devaraj, R., & Hemanandh, J. Design and analysis of exhaust manifold of the spark ignition engine for emission reduction. International Journal of Ambient Energy, vol 41, issue 6 (2020) pp. 659-664.
[14] Bral, P., Tripathi, J. P., Dewangan, S., & Mahato, A. C. CFD analysis of an exhaust manifold for emission reduction. Materials Today: Proceedings, vol 63, (2022) pp. 354-361.
[15] Nouhaila, O., Hassane, M., Scutaru, M. L., & Jelenschi, L. On the Accuracy of Turbulence Model Simulations of the Exhaust Manifold. Applied Sciences, vol 14, issue 12 (2024) p. 5262.
[16] Tang, Y. P. Effects of Race Car's Speed on the Aerodynamic Aspect Using Computational Fluid Dynamics Analysis. Advanced and Sustainable Technologies (ASET), vol 3, issue 1 (2024) pp. 54-61.
[17] Tan, K. M. Computational Fluid Dynamics Analysis on the Road Bike Using Different Flow Models under Extreme Inlet Velocity. Advanced and Sustainable Technologies (ASET), vol 3, issue 1 (2024) pp. 62-70.
[18] Turangan, M. U. R. Numerical Simulation of the Effect of Flow Velocity and Inlet Position on the Pressure Drop in the Exhaust Manifold. Indonesian Journal of Maritime Technology, vol 1, issue 1 (2023) pp. 1-7.

Conflict of interest statement: The author declares no conflict of interest.

Author contributions statement: Conceptualization; Methodology; Software; Formal Analysis; Investigation; Writing & Editing, Vinoth Baskaran.