Journal of Applied Fluid Mechanics, Vol. 19, No. 1, pp. 3280-3296, 2026.
Available online at www.jafmonline.net, ISSN 1735-3572, EISSN 1735-3645.
https://doi.org/10.47176/jafm.19.1.3592

Numerical Investigation using Computational Fluid Dynamics to Improve Thermal Efficiency of Exhaust Manifolds in Hot Air Sterilization Systems

M. Wakid[1], A. Widyianto[2†] and A. Widowati[3]

[1] *Department of Automotive Engineering Education, Faculty of Engineering, Universitas Negeri Yogyakarta, 55281, Indonesia*
[2] *Department of Mechanical and Automotive Engineering, Faculty of Vocational, Universitas Negeri Yogyakarta, 55281, Indonesia*
[3] *Department of Science Education, Faculty of Mathematics and Natural Sciences, Universitas Negeri Yogyakarta, 55281, Indonesia*

†*Corresponding Author Email:* aguswidyianto@uny.ac.id

ABSTRACT

This computational fluid dynamics study examined the exhaust manifold of a hot air sterilizer. The exhaust manifold's thermal and flow properties at varied engine rpm were studied to optimize the temperature profile, pressure regulation, and exhaust gas velocity. Optimizing sterilizer efficiency and heat control requires this evaluation. This study simulated Computational Fluid Dynamics (CFD) using Ansys 2023 (Student Version). Boundary conditions and a structured mesh were employed to construct the manifold for engine speeds ranging from 750 to 2000 rpm. The exhaust manifold heat transfer simulations utilized a k-ε turbulence model to account for flow dynamics accurately. Recorded and assessed exhaust gas velocities, temperatures, and pressure contours to assess system performance. The exhaust flow velocity and pressure increased substantially with engine speed up to 2000 rpm, when the engine reached 70 m/s and more than 2 kPa. The temperature profile showed considerable cooling with a maximum wall temperature of 444°C at the highest engine speed. The study found that manifold geometry, form, and material choice are crucial in limiting back pressure and heat loads at higher engine rpm. Finally, mechanical stasis zones become observable at lower flow velocities, which may help refine the design. A well-designed exhaust manifold is necessary to prevent overheating and ensure optimal gas flow. Materials like steel, which dissipate heat, are key to gadget longevity. The study's findings can enhance the performance and reliability of hot air sterilizers in high-temperature environments.

Article History

Received April 10, 2025
Revised July 13, 2025
Accepted August 23, 2025
Available online November 5, 2025

Keywords:

Computational fluid dynamics
Exhaust manifold
Hot air sterilization
Thermal management
Gas flow analysis

1. INTRODUCTION

The exhaust system controls exhaust gases from a vehicle's combustion chamber. Automobile exhaust system requirements have increased significantly to address concerns about the environmental impact of exhaust gases and to comply with British Standard (BS) pollutant emission rules (Bral, et al., 2022; Haskew & Liberty, 1991). Exhaust velocity and back pressure affect emissions in an exhaust system. Back pressure is necessary to overcome the resistance to the expulsion of exhaust gases from the engine. The pressure, which seems to build within the exhaust manifold system at the point where pipe gas exits, measures back pressure. Emissions must be kept within limits during the engine's operation, especially during high revolutions, and illicit back pressure should set in. Considerations related to the design of the exhaust system and its respective components have improved. Thus, new strategies have been developed to minimize back pressure without compromising performance outputs. The exhaust gas is sometimes pushed out of the engine, and its velocity is referred to as exhaust velocity (Kumar et al., 2022). To ensure that the engine performs optimally and reduces pollution, it is necessary to understand and enhance the speed at which the exhaust gases are expelled. When the exhaust velocity is increased, the scavenging action helps to effectively displace leftover combustion gases from the combustion chamber during the intake stroke. Reduces pollutants and improves engine efficiency. Increased exhaust velocity can cause noise and reduced torque at lower engine speeds.

Higher backpressure levels can strain the engine, require more mechanical effort, and lead to increased fuel consumption, higher emissions of particulate matter (PM), carbon monoxide (CO), and elevated exhaust gas temperatures, as well as potential seal issues. However, an increased exhaust speed may allow for more exhaust gas to be expelled rather than trapped during valve overlap. In other words, while increased back pressure may be detrimental to an engine's performance, achieving the optimum value of exhaust velocity helps control emissions by facilitating the clearance of exhaust gases when the inlet and outlet valves are open or overlapping (Sulistyo, et al. 2023). Modern engine designs focus more on controlling back pressure and exhaust velocity, particularly in relation to turbocharging and supercharging. These systems are critical aspects of the exhaust system dynamics processes, affecting back pressure and velocity. There is a need for reliable and well-tuned exhaust systems to reap the benefits of forced induction in engines without causing adverse effects on the engines and emission problems.

Additionally, exhaust system design also impacts other areas beyond emissions and engine performance. For instance, it also affects the noise level in the vehicle, the comfort inside the cabin, and the driving experience. Hence, engineers are discouraged from taking knee-jerk reactions during exhaust system design and instead resort to the name and form of the system, where such systemic complexities are avoided, to ensure that legislation is observed and vehicles perform properly. Therefore, the balance that must be struck between back pressure and exhaust velocity is a vital detail to consider when formulating exhaust systems, particularly in light of environmental and emission control factors (Sadhasivam, et al. 2021). Changes in materials, manufacturing methods, and the advancement of computational fluid dynamics underline the design of modern vehicles, which in turn includes the contemporary designs of exhaust systems.

Exhaust system Computational Fluid Dynamics (CFD) analysis has been the focus of recent research (Kresovic, et al., 2002; Xu, et al. 2012). The performance and thermal behavior of exhaust manifold systems have been extensively researched under various engine operating conditions. In the work described by Alphonse and Kumar, the authors conducted a CFD analysis of an exhaust manifold to estimate and optimize its thermal performance. Fluid mechanics, heat transfer, and combustion within the manifold have also been studied in detail to maximize effective heat retention and facilitate efficient heat management. The researchers performed CFD simulations to capture the complex flow and thermal profiles in the exhaust system, thereby improving its performance and durability. Another study (Desai et al. 2022) utilized CFD to assess the effects of the manifold geometry, operating speed, temperature, and back pressure on alternative fuel engine exhaust manifolds. The performance measures of the produced exhaust gas system were subject to computer simulations, with operating conditions and fuels influencing the examined system. Computational fluid dynamics studies focused on exhaust manifold flow properties, temperature and pressure distribution, and optimization of its configuration for emission reductions. These studies also consider the engine systems that employ different fuel strategies and find enhancements or penalties in performance.

The work (Pathak & Deshmukh, 2021) reported that exhaust manifolds fitted with fins enhanced the heat transmission rate by 3.01% and decreased the peak temperature by 2.42%. This new approach demonstrates how small modifications in exhaust component design can enhance the thermal characteristics and durability of the parts. Researchers improved heat transfer efficiency in the exhaust manifold by enhancing heat dissipation features, reducing thermal stresses, and increasing the component's durability. Statistical analysis of a four-stroke petrol engine (Teja, et al. 2016) at 2800 rpm with two different exhaust manifolds highlights the importance of design in engine use. The data demonstrate that during various operational regimes of turbocharged engines, the exhaust manifold is designed to ensure maximum efficiency and engine performance. The results aimed at understanding the design aspects of exhaust flow patterns, pressure distribution, and scavenging efficiency through numerical techniques, which assisted engineers in enhancing the performance and fuel economy of the engines.

The research (Bajpai, et al. 2017) investigates exhaust channel development, including its transformation, modeling, computation, and performance using different fuel grades. This work emphasizes the importance of adapting exhaust system configurations to accommodate changes in fuel types and their corresponding emissions. In this effort, researchers experiment with modified exhaust manifold shapes and designs under different fuel conditions to develop more eco-friendly propulsion systems that support existing environmental regulations. Computational fluid dynamics analysis was employed to analyze the flow pressure, velocity, and temperature of the exhaust manifold (Manohar & Krishnaraj, 2018). Advanced numerical methods were applied to study the complex phenomena in the exhaust manifold and target the optimization of flow dynamics and thermal management systems. Such knowledge is applied to enhance exhaust systems that aim for minimum pressure losses, maximum scavenging efficiency, and optimal engine performance. Lastly, the steady speed equivalent technique predicts the consequences of non-steady heat transfer in the thermal management of a vehicle (Guoquan, et al. 2021). Knowledge of flue gas temperature pulses and external airflow is necessary for effective thermal management. Knowing how heat transfer changes over time would enable engineers to configure cooling systems, components, and strategies to improve operational temperatures and vehicle performance while reducing the risk of overheating and structural failures.

Similar to the current research (Maheshappa, et al. 2013), the earlier research performed a CFD analysis of a diesel engine exhaust system to reduce the back pressure effects induced by a catalytic converter by reducing its size. This study analyzed the high-temperature and high-pressure fluid flow and chemical reaction phenomena in the catalytic converter to develop methods for minimizing sized particles, thereby reducing back pressure and enhancing engine performance. CFD enabled researchers to study the interactions between the surfaces of catalysts and exhaust gases, allowing for the design of new catalytic converters with efficient systems for reducing particle size. To minimize emissions, CFD analysis was conducted to optimize the exhaust manifold configuration of a multi-cylinder spark-ignited (SI) engine (Umesh & Rajagopal, 2013). The exhaust manifold configurations and flow designs enabled the researchers to limit exhaust gas recirculation and promote the scavenging process, enhancing emission control. This publication also aims to develop exhaust manifolds that enhance performance while complying with stringent environmental standards.

A specific design of the three-way catalytic converter was studied concerning axial and radial flow (Taibani & Kalamkar, 2012). Researchers studied various flow conditions and converter configurations to optimize the use of the catalyst and enhance conversion efficiency across a broader range of operating conditions. Researchers examined alternative flow scenarios and converter configurations to enhance catalyst use and conversion efficiency under various operating conditions. This detailed study helps us understand the performance of catalytic converters under different flow regimes and improve emissions control strategies. For example, Seenikannan et al. (2008) analyzed Y-section exhaust manifold systems to enhance engine performance. Researchers optimized the Y-section manifold design and flow to minimize pressure losses and enhance exhaust gas scavenging, thereby improving engine performance and efficiency. This study highlights the role of exhaust manifold design in optimizing engine performance across various operating conditions.

Deger et al. (2004) investigated the properties of exhaust manifolds by performing CFD-FE analysis to optimize diesel engines. The researchers utilized computational fluid dynamics and finite element analysis to visualize the exhaust manifold structure and flow, thereby developing designs that can withstand high temperatures and pressures while improving the flow. In another publication (Al-Khishali, et al. 2010), the authors focused on the flow within single-cylinder gasoline engine exhaust manifolds from both experimental and theoretical perspectives. The researchers assessed the manifold exhaust flow and pressure using experimental data and theoretical models, thus improving the engine's performance and controlling emissions. Additionally, a CFD analysis of the exhaust manifold's back pressure was conducted. The study by Gopal, Kumar, and Kumaragurubaran focuses on the CONTESSA engine with 20 HP at 2000 rpm. Engineers recognized the importance of back pressure on the engine's performance. They conducted computer simulations to analyze the working conditions and the layout of the exhaust system. This is a practical example of how computational fluid dynamics aids in designing more effective and cleaner automotive systems by illustrating how the dynamics of the exhaust system's geometry, design, and operational regimes influence the engine's performance and emissions.

Previous studies have adequately addressed the contribution of CFD analysis to automotive exhaust systems, especially in enhancing efficiencies and reducing pollutants. However, relatively little work has been done on optimizing the exhaust manifolds of hot air sterilizers. Most past works have been in automotive-related studies, where diverse studies have analyzed the influence of manifold shape and material properties on flow dynamics and temperature profiles. Some studies on exhaust manifold designs have sought to optimize emissions and the engine's heating system; however, very few of these designs have been geared towards industrial hot air sterilizers, which need high thermal management and uniform heating (Teja et al., 2016).

The use of Computational Fluid Dynamics (CFD) in thermal sterilizing systems powered by exhaust heat has not been extensively investigated, despite its widespread application in automobile exhaust systems to improve engine performance, reduce emissions, and optimize flow efficiency. To deactivate microbes, sterilization chambers must precisely regulate the retention and distribution of heat, unlike traditional exhaust systems. This is because reaching and evenly maintaining a threshold temperature takes time. Using computational fluid dynamics (CFD) methods, this research contributes to the growing body of literature on waste heat recovery by assessing and enhancing the thermal and flow properties of an exhaust manifold that serves as the primary heat source for a hot air sterilizer. For industrial or remote sterilizing solutions that are both cost-effective and fuel-efficient, this research fills a gap in the literature by combining engine exhaust system modeling with thermal process engineering.

This gap in existing studies will be the driver of this study, whose focus will be on the use of CFD modeling to enhance the performance of the hot-air sterilizer exhaust manifold. Using CFD modeling, an attempt will be made to study the thermal and flow characteristics at various engine drive speeds, focusing on exhaust gas velocities, temperatures, and pressure distributions. This study not only builds upon previously carried out studies on exhaust manifold optimization but also delves into a new area of applying these principles to improve the operational efficiency of hot air sterilizers.

According to prior studies, the CFD method remains the most effective for simulating exhaust gas flow. This research evaluates hot air sterilization systems from the exhaust manifold in terms of heat dispersion, exhaust manifold wall heat, and exhaust gas flow velocity at various engine rotation settings, given their widespread and successful application. CFD models exhaust gas flow dynamics in detail and precision, allowing for a thorough understanding of how engine speed affects the thermal and hydrodynamic characteristics of the exhaust system. CFD models enable researchers to investigate the intricate interactions between exhaust gases, manifold walls, and surrounding components, thereby optimizing hot air sterilization instruments for maximum efficiency and effectiveness. CFD analysis also reveals the exhaust manifold's heat distribution, temperature gradients, and thermal stresses on the manifold walls. For these reasons, such knowledge is essential for exhaust systems designed to withstand extremely high temperatures and thermal cycling while maintaining their structural integrity and performance.

Additionally, CFD simulations enable the estimation of exhaust gas flow velocity profiles for various engine operating conditions, allowing engineers to design the exhaust system's shape and layout optimally, minimize pressure losses, and maximize scavenging efficiency. Additionally, the hot air sterilization room temperature can be computed by CFD through the heat propagation from the exhaust manifold by the researchers. Engineers can perform direct uniform heating and sterilization of specific surfaces or materials by utilizing the exhaust manifold temperature profiles in the sterilization chamber's heat transfer process. This systemic and multifaceted view of system design and optimization illustrates CFD's widespread usability and effectiveness in analyzing and designing exhaust systems in various case scenarios, such as cars or industrial hot air sterilization processes.

Previous studies have utilized CFD to enhance engine output, improve turbocharging efficiency, or reduce emissions by optimizing exhaust manifolds (e.g., Maheshappa et al., 2013). This study differs from others in that it focuses on utilizing the exhaust manifold as a heat source for a hot air sterilization system. Sterilization systems require stable thermal loads and uniform high temperatures over time, which differs from most other designs. The goal of this study is not to increase combustion efficiency, but to test and improve the exhaust manifold's ability to transmit heat, flow evenly, and function effectively as a sterilizing instrument under various engine speeds and boundary conditions. There is limited research on this application setting in the CFD literature, and it is a new area where automotive waste heat recovery and thermal process engineering intersect.

2. METHODOLOGY

A fluid domain was created for this study, and ANSYS Fluent (Student Version) software was used to carry out the simulation, as it is well-equipped to handle such complex fluid thermal analyses. The model considered a three-dimensional exhaust manifold, and inlet and outlet boundary conditions were laid out according to the operating state of hot air sterilizers. Thermal conductivity and specific heat capacity were selected as material properties of a stainless-steel exhaust manifold. This type of exhaust manifold is durable and can manage high heat levels.

2.1 Geometry and Mesh Generation

Using the specified geometric parameters, the exhaust manifold was modeled. This geometry is intended to reproduce the exhaust manifold configuration according to the given data however more details can be seen in Fig. 1.

A structured mesh is generated to obtain precise fluid flow and heat transfer analysis. The mesh is coarse in most domains, except in the bend or intersection regions and other areas with steep gradients, to avoid mesh-dependent errors and achieve convergence. For more details, please refer to Fig. 1.

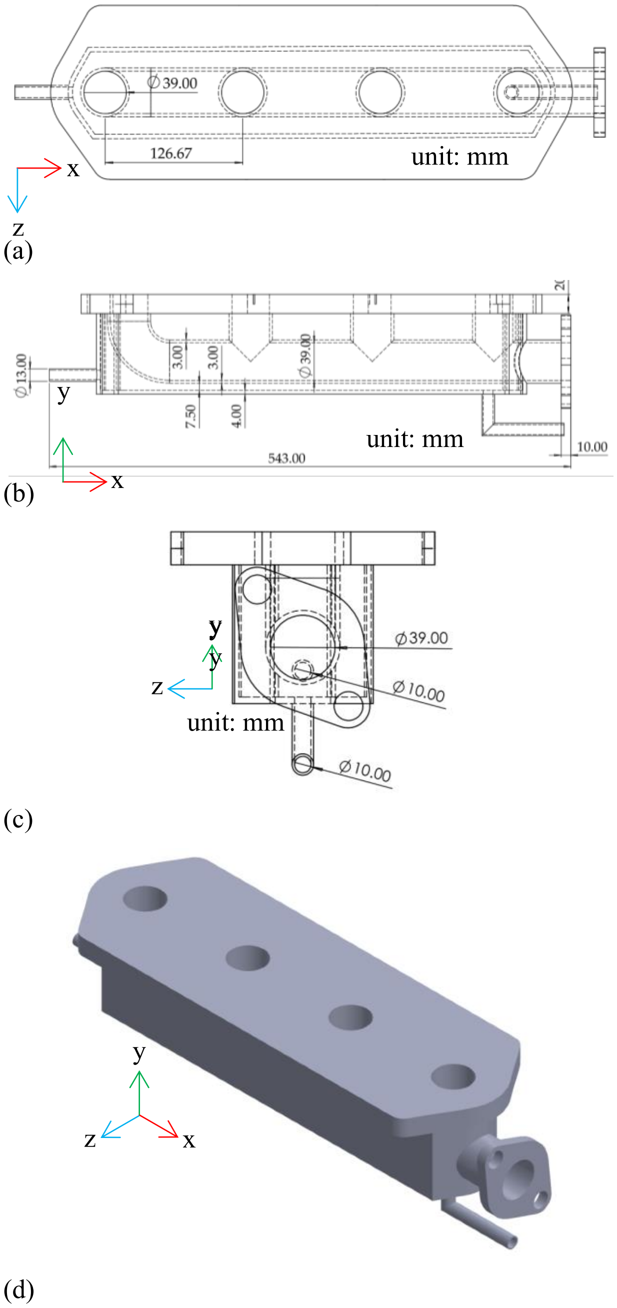
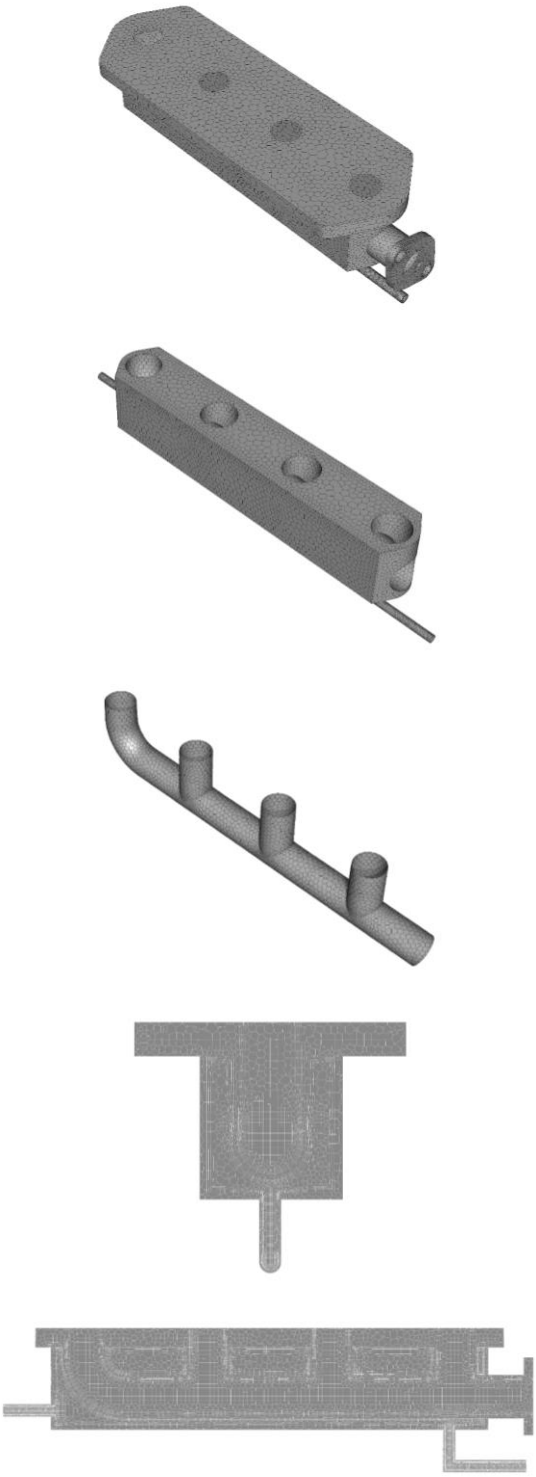


Fig. 1 Dimensions of the exhaust manifold model from a) top, b) right side, c) front, and d) isometric views

2.2 Mesh Setup

The mesh was adapted in terms of resolution to include sufficient details in high-gradient zones, particularly near the inlet and outlet, where temperature and velocity changes are the most rapid. A mesh independence test was conducted with particular emphasis on the chosen mesh size, ensuring that any additional mesh refinement would not significantly alter the outcome, thereby ensuring accuracy at a manageable computational cost. Figure 2 shows the mesh model for the exhaust manifold.



(a)

(b)

(c)

(d)

(e)

Fig. 2 Mesh model exhaust manifold results for air sterilization

Table 1 Grid independence for model exhaust

No	Mesh densities		
	Coarse mesh (element)	Medium mesh (element)	Fine mesh (element)
1	212,000	385,000	676,000

Table 1 summarizes the results of this investigation on grid independence. The engine speed during the simulations was set at 1500 rpm for each mesh. At the manifold outlet, we compared the maximum wall temperatures and exhaust gas velocities that resulted. The results showed that the mesh had converged, with a relative temperature error of less than 1.7% and a relative exhaust gas velocity error of less than 1.2%. For all subsequent simulations, the medium mesh, with approximately 385,000 elements, was used to achieve a compromise between computational efficiency and the accuracy of the results. Without overwhelming computing resources, these values ensure that the mesh density is sufficient to represent flow dynamics and heat gradients accurately.

2.3 Boundary Conditions

Simulation boundary conditions were based on the crankcase rotation speed developments for the four exhausts. The boundary conditions applied were the following. (1) The temperature values at the exhaust manifold inlets were changed during operation in line with the engine revolutions per minute (rpm) classification: 750 rpm, 1000 rpm, 1500 rpm, and 2000 rpm. The temperature values are provided in the table. For example, when the engine was operating at 750 rpm, the temperature of the inlet was adjusted to 200°C for exhaust manifold 1, 185°C for exhaust manifold 2, 170°C for exhaust manifold 3, and 155°C for exhaust manifold 4. (2) The inlet velocity is also changed based on the engine rotation speed and the values shown in Table 2. For instance, when the engine was operating at 750 rpm, a constant inlet velocity of 6 m/s was applied to all exhaust manifolds. (3) The outlet pressure was set to atmospheric pressure at all outlets to facilitate realistic outflow conditions.

Table 2 Boundary condition settings

Engine rotation (rpm)	Intake manifold 1	
	Temperature (°C)	Velocity (m/s)
750	200	6
1000	280	7
1500	390	9
2000	460	13
Engine rotation (rpm)	Intake manifold 2	
	Temperature (°C)	Velocity (m/s)
750	185	6
1000	268	7
1500	380	9
2000	453	13
Engine rotation (rpm)	Intake manifold 3	
	Temperature (°C)	Velocity (m/s)
750	170	6
1000	256	7
1500	370	9
2000	446	13
Engine rotation (rpm)	Intake manifold 4	
	Temperature (°C)	Velocity (m/s)
750	155	6
1000	244	7
1500	360	9
2000	439	13

The differences in inlet temperature and velocity measurements with engine speed were attributed to thermal output correlations in small internal combustion engines operating under light-load conditions. These were changed to show the sterilizer's exhaust heat potential. Using heat balance calculations that took into account the rise in temperature in the combustion chamber, as described in Guoquan et al. (2021), we approximated the rise in temperature with RPM. We used simplified 1D continuity equations to determine the input velocities, assuming that the port diameter remained constant and that the engine RPM increased linearly. We utilized a constant atmospheric pressure output to simplify the simulation setup and enable comparison of engine speeds. However, we recognize that this may not accurately represent the dynamic resistance effects induced by the geometry of the downstream ductwork or sterilizer chamber. In the future, this work will incorporate a non-uniform outlet pressure profile or mass flow boundary condition based on recorded back pressure or pressure gradient, particularly when the sterilizer is operating in a transient state.

2.4 Turbulence Model

Considering its robustness and applicability on numerous internal combustion engine simulations, a standard k-ε turbulence model was chosen. This model will also be efficient enough to simulate turbulent flow inside the exhaust manifolds, which is complex especially at higher engine rotational speeds. This study employs the k-ε (kappa-epsilon) turbulence model, which has been proven useful in predicting flow behavior closer to the surfaces of industrial equipment, such as exhaust manifolds, that experience high turbulent flow.

The standard k-ε model was chosen for this study because it strikes a good balance between computational cost and predictive capability, even though advanced turbulence models like RNG k-ε, Reynolds Stress Model (RSM), or Large Eddy Simulation (LES) are renowned for their improved accuracy in resolving complex flow phenomena like recirculation and swirl. Previous research on exhaust manifold simulations, such as the works by Alphonse and Kumar (2021), Maheshappa et al. (2013), and Manohar and Krishnaraj (2018), has confirmed that the standard k-ε model is sufficiently precise for predicting flow and thermal behavior in comparable internal flow scenarios. Additionally, the standard k-ε model has been extensively utilized in CFD analyses of exhaust systems featuring curved geometries, demonstrating reliable convergence and stable predictions, especially in steady-state simulations involving high Reynolds number flows, as observed in this study. Although models like RSM or LES might be able to capture more intricate turbulence structures, they entail a considerably higher computational expense and were deemed outside the parameters of this initial design optimization study.

2.5 Simulation Parameters

The simulations were performed under steady state to evaluate the thermal and fluid flow performance of the exhaust manifolds at various engine rotation speeds. Each manifold was simulated at different rotation speeds (750 rpm, 1000 rpm, 1500 rpm, 2000 rpm), and the temperature and velocity contours were recorded.

2.6 Convergence Criteria

Convergence was assessed by monitoring the residuals of the relevant equations, which incorporated continuity, momentum, and energy, and checking whether key output parameters —temperature and velocity — remained stable. After a prescribed number of iterations, such that the residuals were less than 1e-4 and there were no changes in the temperature and velocity profiles from one iteration to the next, the solution was considered to have converged.

2.7 Material Properties

In the simulations performed, attention to detail on material properties was undertaken to effectively predict the thermal and mechanical behavior of exhaust system components. The materials used in the study included structural steel, which comprised the bulk of the exhaust manifold, and nitrogen oxides (NOx), which were the primary exhaust gases. The decision to use steel for the exhaust manifold was informed by the fact that steel is a commonly used material in automotive applications, particularly for components subjected to high operating temperatures and mechanical forces. The steel used in the simulations had an overall density of 8,030 kg/m^3, which confirmed its high structural strength. The thermal conductivity was set to 16.27 W/m-K, the appropriate value for the exhaust gases to flow through the manifold walls without causing any hot spots or compromising the manifold's structural integrity. The specific heat of steel was taken to be 502.48 J/kg-K, indicating that the material could retain considerable heat without significant changes in its temperature range, thereby making heat management more effective in the exhaust system. These properties are crucial, considering that the manifold must endure the extreme temperatures typical of high-performance engines.

In the same aspect, nitrogen oxides (NOx) as an exhaust gas were modeled as a gas with a density of 1 kg/m^3. The thermal conductivity of NOx was defined as 0.0454 W/m-K, as it has a low thermal conductivity compared to solid materials, such as steel. The viscosity of nitrogen oxide was taken to be 1.72e-05 kg/m-s, a common value that affects fluid behavior, especially within an exhaust manifold, by causing flow resistance and determining the heat exchange rate. Such material parameters are crucial because they help predict how the exhaust gases will behave when they come into contact with the manifold walls. This prediction is significant because it affects the exhaust system's thermal efficiency and emission performance. All the material properties are listed in Table 3.

The chosen material properties are based on their real-world applications, meaning that the simulation results would be valid and can be used for designing and improving automotive exhaust systems. Again, the different thermal conductivities and specific heats of steel and NOx demonstrate the importance of material selection in ensuring proper thermal management of the exhaust system. On the other hand, the exhaust gas fluid, specifically

Table 3 Material properties of steel and nitrogen oxide

Properties / Material	Density (kg/m^3)	Thermal conductivity (W/m-K)
Steel	8030	16.27
Nitrogen oxides	1	0.0454
Properties Material	Specific Heat (J/kg-K)	Viscosity (kg/m-s)
Steel	502.48	-
Nitrogen oxides	-	1.72e-05

nitrogen oxide (NO), was simulated by considering a density of 1 kg/m³ and a thermal conductivity of 0.0454 W/m·K, which are the standard material properties for this gas. The viscosity of nitrogen oxide at 1.72e-05 kg/m-s was considered. The material property settings play a vital role in accurately modeling the behavior of the exhaust gases within the exhaust manifold. The density and thermal conductivity of nitrogen oxide, an essential component of the exhaust gas, differ from those of steel, which influence the system's heat transfer process.

2.8 Step-by-Step Simulation Process

To provide a full understanding of the exhaust manifold working, the CFD simulation process was in turn broken this way as well: (1) Generation of Geometry, a mesh was created from a 3D CAD model of the exhaust manifold showing internal flow paths of the exhaust gases and other significant geometrical features that would affect the flow. (2) In the mesh generation, the geometry surface was transformed into a parametric volume attached to finite elements. It provided an enhanced mesh around the critical areas to correct the temperature and pressure ratios. The last mesh was confirmed via a mesh analysis that revealed looseness. (3) Applying Border Conditions and Assigning Material Properties, the inlet and outlet boundary conditions were set, and the stainless steel material properties were incorporated into the model to carry out heat transfer analysis through the manifold walls. (4) Introducing the Turbulence Model and Flow Parameters, the k-εpsilon model turbulence was chosen as the best computation effort used against flow simulation accuracy to enable modeling of the exit gas flow inside the manifold. (5) Execution of the Simulation: The simulation was conducted under various operating conditions, including steady and transient conditions without limits, enhanced by excited variations in exhaust gas velocity and temperature. A convergence assessment was performed, and the results were filtered according to acceptable standards. (6) Post-processing of Generated Results: Results on temperatures, velocity, and pressure contours were analyzed to demonstrate the use of the exhaust manifold for virtually every driver operating under those working conditions.

2.9 Justification of Methodological Choices

The adopted methodology enables an effective and accurate thermal and flow analysis of the exhaust manifold. The results can be obtained without straining computing resources due to the incorporation of the k-ε turbulence model in the study. The mesh independence study and the systematic setup of boundary conditions support the consistent and reproducible enhancement of the simulations, and the results are representative of the actual performance.

2.10 Modeling Assumptions and Limitations

This study assumes that the thermal conductivity (16.27 W/m·K) and specific heat capacity (502.48 J/kg·K) of stainless steel remain constant, drawing on average values within the operating temperature range of 300–500°C, consistent with standard engineering references. This simplification is often employed in the initial phases of CFD modeling to lower computational expenses and ensure numerical stability. Recognize that these properties depend on temperature and can change considerably, particularly at temperatures exceeding 400°C. For instance, the thermal conductivity of stainless steel can rise above 20 W/m·K when temperatures go beyond 500°C, and the specific heat capacity also tends to increase. The assumption of constant properties may have a minor impact on the accuracy of thermal gradient predictions in high-temperature areas. In upcoming research will integrate temperature-dependent material property datasets to more accurately reflect the thermophysical behavior of the manifold when subjected to different thermal loads.

This study primarily examines the convective heat transfer that occurs between the exhaust gases and the inner surfaces of the manifold and sterilization chamber, as this process is the most significant under the simulated flow conditions. We recognize that radiative heat exchange, particularly between high-temperature steel walls and nearby surfaces or enclosures, plays a role in the overall distribution of energy. Radiation is especially important when wall temperatures go beyond 400°C, as surface-to-surface radiative losses can greatly influence thermal gradients and heating efficiency. Future studies will take this into consideration by incorporating the Discrete Ordinates (DO) radiation model, enabling directional radiative transfer between surfaces and the surrounding environment. Integrating radiation modeling is anticipated to improve the precision of temperature forecasts, especially in stagnant areas of the sterilization chamber or on the surfaces of external walls. The integrated convective-radiative model will more accurately represent the actual thermal conditions of the system.

3. RESULTS AND DISCUSSION

3.1 Exhaust Manifold Model

The speed and heat ventilation within the exhaust manifold is represented in Fig. 3 at one engine rpm. Due to the way the gases flow from the inlet side to the outlet side of the exhaust manifold, this flow pattern exhibits different temperature zones in various parts of the manifold.

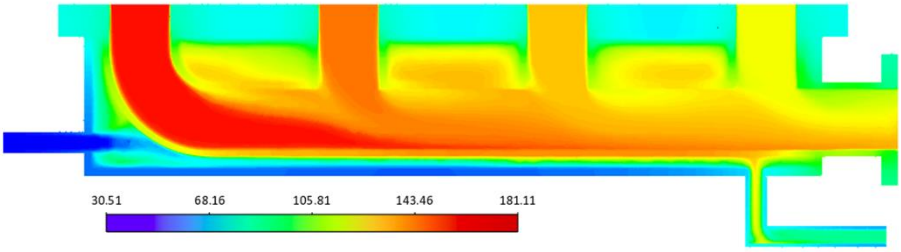


Fig. 3 Flow Distribution within the Exhaust Manifold

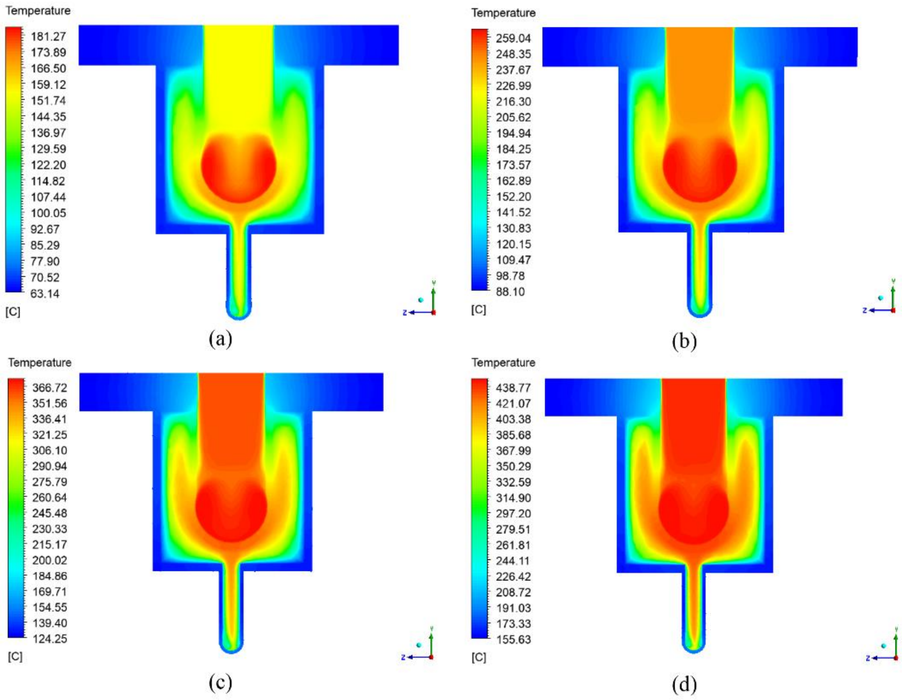


Fig. 4 Temperature Distribution Across Different Engine Speeds

3.2 Temperature Distribution

Considered in this context, the temperature distribution follows the same law, with high temperatures detected in the regions of the exhaust manifold wall, especially in the outlet parts. The temperatures around the manifold indicate that the walls collect heat from hot exhaust gases. The temperature variation from the inlet to the outlet becomes less indicative of the exhaust gases in the manifold, which shows that the exhaust gases are cooling in transit through the manifold. This application of the manifold design is crucial because the heat changes are functioning properly in the manifold, which is essential in protecting the exhaust system from extremely high temperatures. Figure 3 illustrates the necessity to control both the velocity and exhaust gas temperature within the exhaust manifold system. Control of these ensures that the exhaust system effectively provides the exhaust manifold's function of controlling temperature and gas flow.

The configurations of the exhaust manifold temperatures at different engine rotational speeds are illustrated in Fig. 4, as shown in the corresponding subfigures (a), (b), (c), and (d). The pictures illustrate the progression of temperature contours as the engine speed increases. Figure 4a (750 rpm), when considering the engine speed of 750 rpm, the temperature inside the manifold does not get too high; rather, the hottest zones are recorded at the center parts of the exhaust manifold. The temperatures rise to around 181.27 degrees Celsius, and the flow is also relatively steady. This suggests that low rotational speeds do not place significant thermal stress on the exhaust system, from which heat emanates relatively uniformly through the manifold. Figure 4b (1000 rpm), when the engine speed rises to 1000 rpm, which is still in the lower band, the temperature rises sharply, where the highest point is recorded at 259.04 degrees Celsius. Unlike before, the area of the high-temperature region also increases in extent over the manifold. This implies that the engine operating at a higher speed also generates more heat that the manifold must evacuate for the engine to work efficiently without overheating.

Figure 4c (1500 rpm): In the case of the engine rotating at 1500 rpm, a temperature of 396.12 °C is recorded. As the heat stress increases, the heat becomes concentrated in the exit region. This further means that the manifold must operate within an even greater temperature range to prevent the collapse of the exhaust system's components. Figure 4d (2000 rpm), at a maximum engine speed of 2000 rpm, the highest manifold temperature is 439.77°C. The heat distribution is now highly localized towards the outlet, indicating the extent to which the thermal stress geometry of the manifold is subjected at higher engine speeds. Therefore, a manifold's heat capacity, thermal content, and other factors become critical in sustaining the loads, especially since most exhaust manifolds must endure extremely high temperatures for prolonged periods.

These temperature profiles are useful in assessing the performance of the exhaust manifold at different engine operating conditions. As the engine speed increases, so do the temperature and the thermal gradients, especially at the outlet sections of the exhaust manifold. High temperatures at these elevations, combined with high engine rpm, suggest that an effective thermal management system must be implemented to prevent performance losses in the exhaust manifold and enhance its durability. The CFD analysis shows that although the exhaust manifold's performance is satisfactory at lower engine speeds, the higher engine speeds that result in high temperatures need materials that can effectively dissipate heat (such as steel) and high thermal conductivity materials. Moreover, the geometry of the exhaust manifold is also conducive to controlling the flow and heat transfer, ensuring that there are no hot spots and that the exhaust manifold can sustain thermal stresses during high-speed operations.

Lessons from Figs. 3 and 4 can be applied to improve the design of the exhaust manifold in the future. One such factor is ensuring that the exhaust manifold material can withstand the extreme heat generated, especially when the engine speed is high. One or several approaches, such as using materials with better thermal conductivity followed by a redesign of the exhaust system, carefully considering the thermal stresses involved, should help avoid exhaust system thermal fatigue. Furthermore, the observed temperature distributions also indicate the need for an optimal combination of exhaust gas flow and cooling. When designing the geometry of the exhaust manifold, this helps improve flow efficiency and heat transfer, contributing to enhanced performance and reduced emissions while maximizing the volume velocity of the gas passing through the exhaust manifold.

3.3 Wall Temperature Distribution in the Exhaust Manifold

As depicted in Fig. 5, the spatial distribution of wall temperature along the exhaust manifold is shown at different engine speeds. The wall temperature plays a significant role in the thermal efficiency and longevity of the exhaust manifold, as excessively high wall temperatures can cause damage to the materials, thereby reducing their strength over time.

The present work also examines the changes in the wall temperature profiles based on the engine of different speeds (i.e., 750 rpm, 1000 rpm, 1500 rpm, and 2000 rpm) by considering four different exhaust manifold designs. Figure 6a (750 rpm), in this engine scenario, the wall temperature peaks at 181.11°C and is fairly uniform across the walls of the manifold. It is also safe to say that the exhaust system body does not suffer from thermal shock or high thermal fatigue, which means it can be considered that the exhaust manifold is working satisfactorily at low engine speeds. Figure 6b (1000 rpm), the maximum temperature increases to 258.81°C as engine speed rises to 1000 rpm, and thermal stratification is more pronounced. The nearer the outlet, the more intense the heat load, meaning the walls in this region carry more heat. This change highlights the problem of improving heat dissipation as the engine speed increases.

Figure 6c (for 1500 rpm), at this (1500) rpm, the wall temperature reaches the maximum value of 369.11 °C. This effect becomes more pronounced as the temperature builds up along the length of the exhaust manifold outlet section. The properties of the exhaust manifold material become more critical at this point as thermal stresses increase, and such material must be above its deterioration or failure point for the application. In Fig. 6d (for 2000 rpm), at this speed (2000 rpm), the exhaust manifold wall temperatures reach their peak, rising to a high of 444.66 °C. Such extreme thermal conditions suggest that a significant amount of heat has transferred to the exhaust manifold walls, particularly near the outlet. The findings further indicate that materials used in the production of the exhaust manifold must have excellent thermal conductivity and good heat resistance to withstand such conditions.

The data represented in Figs. 5 and 6 can be used to evaluate the thermal performance of the exhaust manifold at different engine speeds. As the engine speed increases, the thermal load borne by the manifold increases appreciably, and the region near the outlet suffers more significantly. The highest temperatures attained at 2000 r/min reiterate that the material used to construct the exhaust manifold must withstand extreme thermal conditions without losing its mechanical strength. Steel was thus applied in the present work due to the high need for thermal conduction materials. The conductive

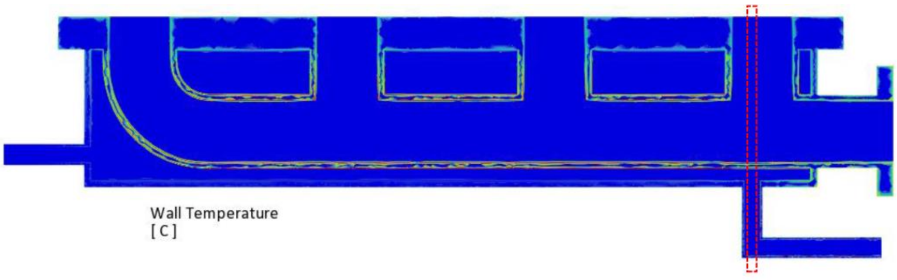


Fig. 5 Wall temperature distribution in the exhaust manifold

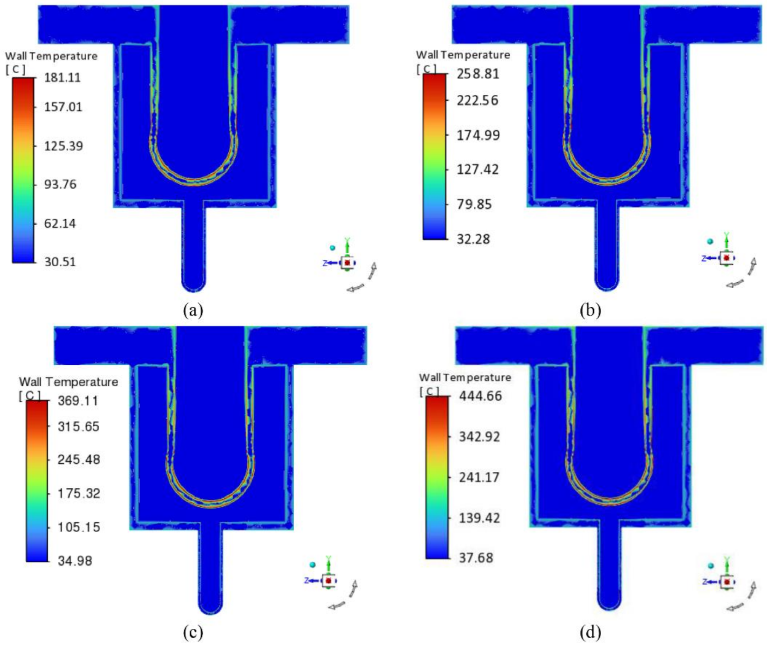


Fig. 6 Temperature profiles at different engine speeds

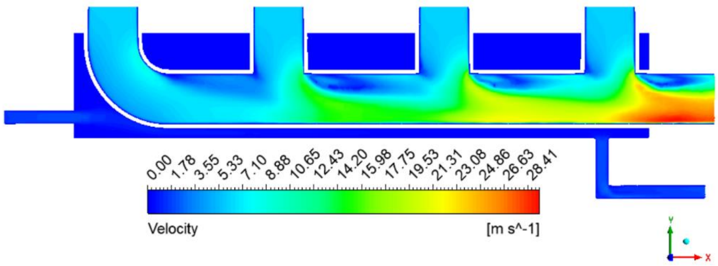


Fig. 7 Velocity profile

property of steel (16.27 W/m-K) promotes rapid heat dissipation, thereby averting excessive temperature rises in localized regions, which helps prolong the exhaust system's lifespan. Apart from that, the material has a high specific heat capacity of 502.48 J/kg-K, which helps absorb and distribute thermal energy, thereby assisting the manifold in withstanding high temperatures.

The temperature profiles in Figs. 5 and 6 serve to highlight important aspects of the exhaust manifold design. The increasing thermal load at higher engine speeds necessitates the use of materials with improved thermal and mechanical properties, along with a modified manifold that evenly disperses heat. Therefore, available designs can aim to enhance the rate of airflow and heat dissipation by modifying such geometrical aspects as the inclusion of cooling fins or increasing the area of the exhaust manifold walls. In addition, the findings also draw attention to the need to control the thermally damaged zone at the outlet of the exhaust manifold, which is exposed to the highest thermal load. However, the performance and reliability of the exhaust components are again improved by avoiding the creation of hot areas and by implementing a more effective heat distribution design.

3.4 Velocity Profile and Flow Analysis

Detailed streamline and vorticity evaluations (see Fig. 7) reveal the presence of recirculation zones and stagnation pockets, particularly around the curved junctions and outlet expansions of the exhaust manifold. These phenomena occur due to boundary layer separation, sudden changes in the direction of flow, and abrupt changes in the cross-sectional area. All of these things mess up the flow of exhaust gas and its momentum. The creation of these low-velocity zones indicates that the flow channel isn't functioning aerodynamically as efficiently as it could be, often resulting in energy loss and heat buildup in certain areas. When it comes to hot air sterilization, these flow problems are very important. Because of the unequal velocity distribution, convective heat transfer is also uneven. This means that certain parts of the sterilization chamber may not get enough heat, while others may get too much. These kinds of problems make sterilization less reliable because killing microbes depends not only on obtaining the right temperatures but also on maintaining them consistently throughout all areas for a certain amount of time. If heat doesn't reach a sufficient temperature, heat-resistant spores may survive. On the other hand, if temperatures are too high in one area, they can hurt delicate materials or parts inside the chamber.

Additional analysis of the vorticity contours reveals regions of strong rotational flow that function as thermal traps. These vortices can create thermal hotspots by holding back high-temperature gases, which extends the duration of heat in specific areas. As time passes, this could lead to thermal stress buildup on the nearby walls, potentially shortening the lifespan of the manifold and downstream components due to material fatigue or oxidation. These findings indicate that there is a need for improved flow control strategies from a design optimization perspective. Reducing sudden geometric changes, enhancing bend radii, and refining inner wall surfaces are fundamental yet impactful strategies to decrease separation and optimize flow paths. Moreover, techniques for flow homogenization, such as the use of guide vanes, baffles, or flow straighteners, can help achieve a more uniform distribution of velocity profiles. Modifying the chamber to include gradual expansions and carefully controlled divergence angles could help reduce vortex formation and improve thermal uniformity.

3.5 Exhaust Gas and Wall Temperature vs. Engine Rotation Speed

Figure 7 depicts the dependence of engine speed (rpm) on the temperature profile of exhaust gases and the exhaust manifold wall, specifically the maximum and minimum gas temperatures, as well as the maximum temperature of the exhaust manifold wall. The curve depicts the construct where, with an increase in engine rotation, the exhaust gas and the manifold wall temperatures also rise. The red line represents the maximum temperature of the exhaust gases as the engine's rotational speed increases. As shown in the graph, the temperature of the exhaust gases increases with every rise in engine rotation speed, reaching more than 500°C at 2000 rpm. This is understandable, as the faster the engine rotates, the more combustion heat is produced, thereby raising the temperature of the combusted gases expelled through the exhaust system. The sharp rise in temperature between the engine speeds of 1500 rpm and 2000 rpm highlights the extreme thermal stresses that high engine speeds impose on the exhaust system. Keeping such high temperatures in check is important, as otherwise, the engine and exhaust system elements would be working under very adverse conditions, causing their performance to deteriorate.

The blue line highlights the maximum exhaust manifold wall temperature. Similarly to the exhaust gas

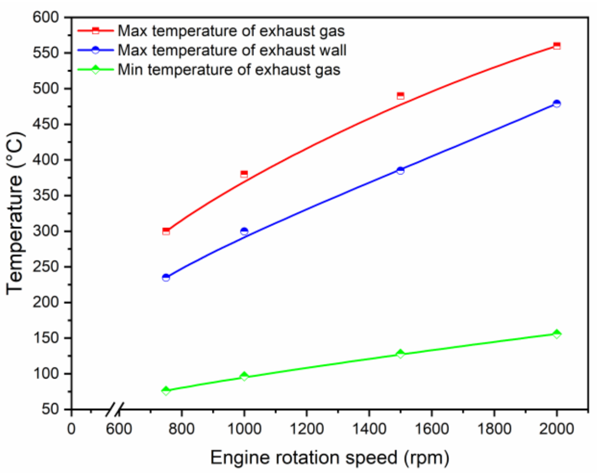


Fig. 7 Exhaust gas and wall temperature vs. Engine rotation speed

temperature, the wall temperature increases with a corresponding rise in engine speed. However, the increase is not as pronounced as that of the gas temperature wall. The wall temperature at 2000 rpm is approximately 400°C. The difference in temperature between the exhaust gas and the wall indicates that the material used for the exhaust manifold, in this case steel, has good heat transfer properties. This proves that the material has a high thermal conductivity, allowing it to cope with the primary thermal mass imposed by the hot exhaust gases and avoid excessive heating.

The green line on the graph represents the minimum temperature recorded for the exhaust gas. The correlation between the minimum temperature and engine speed is positive; however, the minimum temperature would always be much less than the maximum temperature measured. When the engine runs at 2000 rpm, the minimum exhaust gas temperature is slightly less than 200°C. This implies that, even though the exhaust system is exposed to hot gases, there are parts of the manifold where the gas cools more quickly. This temperature difference, apparent externally, indicates that the exhaust manifold is designed to allow heat convection from the gases inside, ensuring that the engine's critical components do not suffer excessive heat stresses.

The data presented in the graph corroborates that engine rotation speed directly affects the thermal parameters of engine components, such as the exhaust duct and the exhaust gases. At higher RPM, the temperatures of the exhaust gases and the walls of the exhaust manifold increase correspondingly. The temperature of the gas is always higher than that of the wall. This shows that the materials and thermal design are significant when designing exhaust systems. Thermal Management: A considerable amount of heat is dissipated, as indicated by the difference between the maximum temperature of the exhaust gas and that of the walls, which helps protect the manifold. The steel described in this article has high thermal conductivity and heat capacity, enabling it to cope with very high temperatures without accumulating excessive internal heat. This also means that, in real life, the components in the exhaust system, such as the exhaust manifold, can withstand very high RPM engines without the possibility of the material distorting or melting due to overheating.

The rise in the exhaust gas temperature with increased engine speed also emphasizes the necessity for better cooling and heat dismissal provisions in performance engines. As engines speed up, internal components are subjected to higher thermal loads. Therefore, there is a need for improved designs to prevent or minimize the occurrence of thermal issues. Inadequate thermal management can lead to excessive temperatures, internal engine knocking, increased emission levels, and reduced fuel efficiency. Based on the information shown in Fig. 7, several important considerations for the development of future exhaust systems are evident: (1) Materials, the performance of the exhaust manifold in terms of heat dissipation is a function of the materials used. In applications where high performance is required, the inclusion of high thermal conductivity materials may prove more beneficial, especially at engine revolutions exceeding 1500 rpm, which are characterized by high thermal loads. (2) Thermal barriers, in regions with a low minimum exhaust gas temperature, may be more advantageous if incorporated to maintain the temperature for efficient catalytic conversion of emissions, thereby controlling the gases more effectively. (3) Exhaust manifold shape, the use of exhaust manifold, and control of the geometry of the exhaust streams help create an even flow of exhaust gases and, therefore minimize temperature stresses, hence eliminating the development of permanent hot zones, which too much stress may lose integrity of material subjected over a period.

3.6 Exhaust Gas Velocity and Pressure vs. Engine Rotation Speed

In Fig. 8, we correlate the engine rotational speed (in rpm) with the velocity of the exhaust gas (both at the highest and lowest levels) and the peak exhaust gas pressure within the exhaust manifold. The observation from the graph indicates an increase in exhaust dynamics containment with rising engine speed. Additionally, it highlights the key features of the exhaust system, particularly in terms of velocity and pressure phenomena. The cyan line depicts the maximum exhaust gas velocity at various engine speeds. With the increase in engine rotation speed, the exhaust gas velocity also increases, reaching approximately 70 m/s at 2000 rpm. The increase in velocity is logical, as it relates to the fact that more exhaust gases are generated by an engine at high speeds, which pass through the exhaust manifold at a higher rate. The very steep incline in the velocity curve beyond 1500 rpm highlights the crucial role of exhaust flow management in preventing back pressures, especially in high-revving engines where scavenging efficiency must be optimal to be effective.

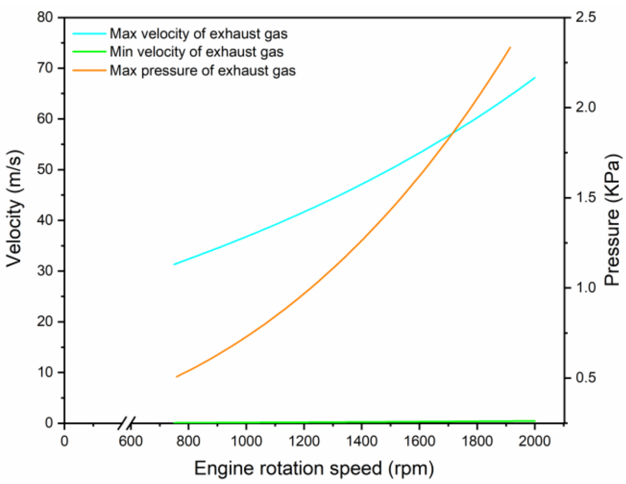


Fig. 8 Exhaust gas velocity and pressure vs. Engine rotation speed

The green line shows the minimum exhaust gas velocity over the range of engine speeds. However, it is rather curious that the minimum velocity remains near constant values, close to 0 m/s over the entire range, which means that sections in the manifold exist where the velocity of the exhaust gases is considerably low or almost at rest. This may happen, for instance, when the flow is recirculated or close to the internal surface of the exhaust manifold on its walls, where the smooth flow is distorted. The velocity of the flow is reduced in the so-called boundary layer. While such a stagnant flow is unlikely to affect performance at low engine speeds due to the relatively high volume of gases produced and injected into the cylinders, at higher engine speeds, such scenarios would be detrimental. Imbalanced flow distribution would ensue, hence the need for design constraints.

Concerning engine speed, the orange line shows the back pressure of exhaust gas as influenced by the maximum engine speed. For engine speed, the pressure rises steeply, increasing from approximately 0.5 kPa at 750 rpm to over 2 kPa at 2000 rpm. This is attributed to back pressure increases due to higher volumes of exhaust gases being pushed through the exhaust manifold. Elevated back pressure levels can deteriorate the engine's performance by impairing the exhaust stroke Wilks. This again increases fuel consumption and contributes to higher gas emissions. Therefore, keeping the pressure developed in the exhaust manifold under control is advisable for optimal engine operation at higher speeds.

Additionally, the graph illustrates how the exhaust system operates with respect to engine load. As the load increases, engine speed also increases, causing a significant change in the velocity and pressure of the exhaust gases, particularly above 1500 rpm. This has several important implications: (1) Velocity and Flow Management, as the exhaust gas velocity increases sharply, the design of the exhaust system must account for increased flow in the system under accelerating engine speeds, especially during maximum revolutions. Systemic failures such as flow separation, turbulence, and back pressure generation, which are detrimental to engine functionality, may occur if these high velocities are not appropriately managed. Proper design of the exhaust manifold, such as gradual changes in optimum axial and radial dimensional ratios, would help to sustain the laminar flow of exhaust fluids thus reducing pressure losses. (2) Back Pressure and Pressure, as exhaust gas pressure rises in an exponential curve, indicate the difficulties of controlling back pressure in high-performance engines. If there is too much back pressure, the engine would have to work harder on the exhaust stroke, which would be counterproductive as it would increase mechanical losses, fuel consumption, and emissions. Thus, the exhaust system, primarily the exhaust manifold, must be designed to minimize or eliminate flow resistance by adjusting the profile and dimensions of the exhaust manifold, as well as the arrangement, to facilitate the flow of exhaust gases and relieve pressure. (3) Minimum Velocity Issues: The minimum velocity of nearly zero registered throughout the range of engine speeds indicates that parts of the exhaust manifold where exhaust gas does not flow exist. This may cause some problems, such as incomplete scavenging of burnt gases, hot spots, or even poor performance of catalytic devices for exhaust cleaning. To eliminate these problems, the hypothesis is that the exhaust manifold must be modified in terms of design to achieve even flow within targeted regions of concern, such as recirculation or stagnation.

The results obtained in Fig. 8 reveal some key implications for the design of the exhaust manifold, in particular: (1) Enhancing Flow Path Characteristics, the increase in flow velocity within the exhaust manifold at greater engine speeds, as clearly shown in the graph above why the appropriate design of the flow path must be employed within the exhaust manifold system. Exhaust manifolds need to be designed in a way that eliminates sharp bends, sudden enlargements, or reductions that would hinder flow and promote high back pressure. (2) Material Selection for Pressure Criteria: Considering the increased engine speeds and pressures within the exhaust manifold, the material used to make the exhaust manifold should be strong enough to withstand the added pressures. This may involve using high-strength, temperature-resistant materials for the manifold to remain well-preserved despite the high pressure. (3) Performance Enhancing Tuning of the Exhaust System, the study's findings also assert that race exhaust systems, particularly on high-power engines, should be performance-tuned to facilitate an appropriate exhaust pressure and speed. Some of the methods of exhaust tuning that can be used to achieve the necessary limits for optimal engine efficiency include adjusting the lengths of the inlet and exhaust manifolds, positioning the catalytic converter strategically, and utilizing exhaust scavenging techniques (for example, pulse tuning).

3.7 Flow Distribution Analysis

The CFD analysis of the exhaust manifold was conducted at various engine rotation speeds (750 rpm, 1000 rpm, 1500 rpm, 2000 rpm), focusing on the velocity, temperature, and pressure distributions within the exhaust manifold. The results indicate significant variations in flow dynamics across different exhaust manifolds. At lower engine speeds (750 rpm), the velocity distribution across all four exhaust manifolds was relatively uniform, ranging from 6 to 7 m/s. However, as the engine speed increased to 2000 rpm, the velocity within the exhaust manifolds showed a marked increase, reaching up to 13 m/s. This increase in velocity is crucial for efficient scavenging of exhaust gases, particularly at higher engine speeds, where rapid expulsion of gases is necessary to maintain engine performance.

Temperature distributions within the exhaust manifolds followed a similar trend, with temperature increasing as engine speed increased. At 750 rpm, the temperatures ranged from 155°C to 200°C, whereas at 2000 rpm, the temperatures peaked between 439°C and 460°C. The higher temperatures observed in exhaust manifolds 1 and 4 suggest that these configurations are more effective at retaining heat, which could potentially enhance the thermal efficiency of the exhaust system. However, the corresponding increase in pressure, particularly at higher engine speeds, indicates a need for careful management of back pressure to avoid adverse effects on engine performance. The pressure distribution analysis revealed higher engine speeds increased back pressure within the exhaust manifold. This is consistent with the expected behavior, as the exhaust gases are expelled more quickly, leading to a buildup of pressure within the system. If not appropriately managed, this pressure increase could lead to higher fuel consumption and increased emissions, emphasizing the need for optimized manifold design to balance flow dynamics and pressure levels.

3.8 Thermal Performance

The thermal performance of the exhaust manifold was assessed by evaluating the temperature distribution and heat transfer efficiency at various engine speeds. The results show that the manifolds effectively manage thermal loads, with the steel material properties playing a significant role in heat dissipation. The high thermal conductivity of steel (16.27 W/m-K) ensures efficient heat transfer from the exhaust gases to the exhaust manifold walls, preventing localized overheating and maintaining the structural integrity of the exhaust manifold. At 2000 rpm, the temperature within the exhaust manifold reached approximately 460°C in Exhaust Manifold 1. Despite the high temperatures, the exhaust manifold maintained consistent performance without signs of thermal degradation. This demonstrates the effectiveness of the material properties and the manifold's design in managing thermal loads, which is critical for long-term durability and performance.

The thermal efficiency of the exhaust manifold was further analyzed by comparing the temperature gradients across different manifold designs. Exhaust Manifold 1, which exhibited the highest temperature, also demonstrated superior heat dissipation capabilities, likely due to its optimized shape and material properties. This suggests that strategic design choices can significantly enhance the thermal performance of exhaust systems.

3.9 Impact of Design Parameters

The analysis of design parameters, such as exhaust manifold shape and material, revealed significant impacts on the performance of the exhaust system. The shape of the exhaust manifold influences the flow dynamics and temperature distribution, with more streamlined designs promoting efficient gas flow and reducing the likelihood of hot spots. For instance, Exhaust Manifold 4, which features a more aerodynamic design, showed lower temperature retention than Exhaust Manifold 1, suggesting a trade-off between thermal retention and flow efficiency. This highlights the importance of optimizing exhaust manifold geometry to balance thermal management and gas flow. Material selection also plays a crucial role in performance. Using steel, with its high thermal conductivity and specific heat capacity, ensures that the exhaust manifold can withstand the high temperatures generated by the exhaust gases while efficiently dissipating heat. This is particularly important in high-performance engines, where the exhaust system must manage significant thermal loads without compromising structural integrity or performance. Summary of the results at different engine speeds as shown in Table 4.

Table 4 Summary of key simulation results at different engine speeds

Engine speed (rpm)	Max wall temp (°C)	Avg outlet velocity (m/s)	Peak pressure (kPa)
750	181	28.4	1.64
1000	259	33.1	2.03
1500	369	42.6	2.31
2000	444	61.4	2.52

3.10Comparison with Literature

The temperature and pressure distributions reported in this research are consistent with the results of other computational fluid dynamics studies conducted on exhaust systems in industrial settings and vehicles in general. Bral indicated the same patterns in the temperature distribution, with the highest temperature values near the inlet and a progressive temperature drop along the manifold (Bral et al., 2022). The temperature drop was due to heat loss by conduction and convection. Deger also noted such effects in their study of exhaust system analysis using CFD and FE techniques, and drew attention to the proper selection of materials to control thermal stresses (Deger et al., 2004).

According to Kumar, high-speed exhaust jets are notoriously destructive when flowing in reverse into the manifold, leading to back pressure that compromises engine performance and manifold life (Kumar et al., 2022). The results of this study support this, as regions of stagnation in angles and corners were observed, which also imply potential thermal accumulation and deterioration of the materials in those regions. Similarly, research conducted by Sadhasivam on two-cylinder exhaust manifolds also clarifies that recirculation zones can cause the formation of hot spots, eventually weakening the material and decreasing the life of the manifold (Sadhasivam et al., 2021). Additionally, works by Teja et al. (2016) and Sadhasivam et al. (2021) elaborate on the need for gradual approaches in the manifold to minimize the resistance to the flow of gases. Such information also aligns with our observations, which suggest that gradual curvature and well-designed geometry can reduce pressure losses and enhance flow.

3.11Implications for Manifold Design Improvements

According to the findings of the CFD analysis, suitable design modifications can be proposed to enhance the performance of the exhaust manifold in hot air sterilizers. The temperature contours indicate that materials like stainless steel, which have moderate thermal conductivity, could be effective. On the contrary, as Sadhasivam et al. (2021) believed, the published outcome states that ceramic-coated steel can enhance thermal resistance without affecting the material's strength, indicating that the materials of some parts could be improved for certain designs. Concerning the design of the manifold, Teja et al. (2016) pointed out the importance of flow path configuration in reducing backpressure and controlling flow separation. Our assessment highlights the benefits of smoother transitions and avoiding sharp turns. This problem can be alleviated by design changes, which help reduce thermal stresses and thus increase the service life of the manifold by eliminating areas where exhaust gases can stagnate or swirl.

According to the research (Alphonse & Kumar, 2021), computational fluid dynamics can be used to enhance the geometric design of complex exhaust systems, such as exhaust manifolds. This is an ideal application because it can shorten design time by allowing designers to create and benchmark the effectiveness of multiple design iterations, thereby improving the exhaust manifold's functionality in terms of fluid flow. CFD results showed that the exhaust manifold thermal load increases with the increase in engine speed. This can be illustrated by the fact that at the highest engine speed of 2000 rpm tested, the exhaust gas temperature exceeded 500 degrees centigrade while the temperature of the manifold wall stood at close to 400 degrees centigrade (Assi, et al. 2020; De Angelis & Palomba, 2004; Li, et al., 2012; Xu et al., 2012). It indicates that such operational conditions require the use of materials with high thermal conductivity and heat resistance, as well as high strength, such as steel for the manifold, to survive without distorting the stress levels imposed during engine operation. The results also emphasized the importance of design solutions that would avoid the so-called hot spots and enhance heat transfer in the manifold. Uneven thermal stresses, resulting from the presence of local hot areas and temperature gradients, may cause the system to fail before its expected lifespan. Consequently, future design changes may also consider enhancing airflow and heat loss by adding cooling fins and/or increasing the surface area to achieve a more uniform temperature distribution.

3.12Practical Relevance and Future Directions

The finding highlights CFD's ability to analyze the exhaust system's flow mode and bring out the significance of that ability beyond automotive to sterilization and industrial heating. "Simulation-based design is another design method highlighted by (Deger et al. 2004; Sadhasivam et al., 2021) which is less expensive than a purely experimental design approach, especially when the experiment is prohibitive or hard to carry out." Future studies may explore additional material options and confirm the CFD findings through actual testing. The literature emphasizes the importance of experimental validation, particularly using materials that can withstand high temperatures, to enhance the reliability of simulation findings substantially (Prithvi et al., 2020; Yogesh, et al. 2020). Combining physical test-out processes and sophisticated simulation programs would result in design optimizations that cut down material costs and operational running costs. Lastly, this research supports the benefits of existing equipment design, which aims to incorporate smart technology into equipment design (Deubert, et al. 2024; Lin et al., 2021; Rodríguez, et al. 2020). Installing machine learning algorithms can enhance the exhaust system's performance by making it more adaptable to operating conditions, thereby prolonging its lifetime and improving energy efficiency.

The present research presents a computational fluid dynamics (CFD) model of a thermal-fluid exhaust manifold integrated into a hot-air sterilisation chamber. Although the model has not yet been validated against real-world data, the numerical results provide valuable insights into temperature changes, flow, and potential design improvements. Things may not behave exactly as predicted in simulations and in reality due to factors such as environmental heat loss, changes in material characteristics at high temperatures, and manufacturing tolerances. This is now being addressed by creating a physical model. Take readings of the wall temperatures using type-K thermocouples and the exhaust velocities at key locations along the outlet duct using hot-wire anemometry. To validate and enhance the simulation model, utilize these measurements in future tasks. Although the authors acknowledge that validation is necessary to ensure the simulation's quantitative accuracy, it is now being used as a predictive design tool.

This study only examined steady-state CFD simulations; however, we know that transient flow events, such as engine start-stop cycles, speed ramps, and temperature oscillations, can have a significant impact on how flow behaves and its thermal stability. These effects are significant in sterilization, where a quick drop in temperature or uneven heating during transitions can make the process less reliable. Additionally, the lack of experimental validation makes it difficult to determine if the model is accurate. To address this, we are currently testing prototypes that utilize built-in thermocouples and flow sensors to obtain real-time data on the temperature of the walls and the speed of the exhaust gases. This will be used to check the CFD model in both steady and unsteady states. Future research will add transient boundary conditions and time-resolved flow analysis to the simulation framework. This will make the model more reliable for real-world use.

4. CONCLUSION

The investigation in question has examined the Computational Fluid Dynamics (CFD) of a hot air sterilizer's exhaust manifold at various engine speeds. The following analysis provides the findings of such a study:

1. Combustion gas flow distribution and velocity. The increasing engine speed accelerates the velocity of the combustion exhaust gases. The maximum velocity of the exhaust gases was recorded at 70 m/s, corresponding to an engine speed of 2000 rpm. Flow patterns indicate the effective expulsion of exhaust gases at rotational velocities of the engine; however, near the manifold walls at low rotational velocities, some regions exhibit stagnant flow.

2. The highest wall temperature recorded in the simulation was approximately 444°C, which remains below the creep strength limit for widely used stainless steels such as SS 304 and SS 316. The materials maintain their structural integrity at temperatures ranging from 500 to 550°C while under normal operating stresses, demonstrating the thermal appropriateness of the chosen material.

3. The simulated exhaust pressure reached a maximum of approximately 2.3 kPa, which is well within the mechanical limits of the thin-walled exhaust tube (yield strength > 200 MPa). This means that the structure can safely handle the thermal-fluid loads without breaking or getting tired.

4. The results of this study could lead to the creation of compact and fuel-efficient sterilization systems that utilize vehicle exhaust or stationary combustion engines. This could be particularly useful in remote medical facilities, food processing units, and waste treatment operations where access to electrical heating is restricted. The findings also contribute to the broader field of industrial waste heat recovery, offering an affordable means to enhance energy efficiency in thermal processes.

ACKNOWLEDGEMENTS

This research was funded by a grant from the Directorate of Research and Community Service, Universitas Negeri Yogyakarta, with contract number T/6.1.29 /UN34.9/PT.01.03/2023.

CONFLICT OF INTEREST

The authors declare that they have no conflict of interest.

AUTHORS CONTRIBUTION

Muhkamad Wakid: Conceptualization, Methodology, Funding acquisition; Supervision; Resources; Conceptualization – Ideas. **Agus Widyianto:** Software, Data Curation, Formal analysis, Writing-Original Draft, Visualization. **Asri Widowati:** Supervision, Writing - Review & Editing

REFERENCES

Al-Khishali, K. J. M., Mashkour, M. A., & Omaraa, E. S. (2010). Analysis of Flow Characteristics In Inlet And Exhaust Manifolds of Experimental Gasoline Combustion In A VCR Engine. *Engineering and Technology Journal, 28*(7). https://doi.org/10.30684/etj.28.7.12

Alphonse, M., & Kumar, R. (2021). Investigation of heat dissipation in exhaust manifold using computational fluid dynamics. *International Journal of Ambient Energy, 42*(9), 999-1004. https://doi.org/10.1080/01430750.2019.1583130

Assi, W. N., Ali, M. A. N., & Allawee, A. S. (2020). Experimental and numerical assessment of a multi-cylinder engine exhaust manifold. IOP Conference Series: *Materials Science and Engineering, 745*, 012071. https://doi.org/10.1088/1757-899X/745/1/012071.

Bajpai, K., Chandrakar, A., Agrawal, A., & Shekhar, S. (2017). CFD analysis of exhaust manifold of SI engine and comparison of back pressure using alternative fuels. *IOSR Journal of Mechanical* *and Civil Engineering, 14*(01), 23-29. https://doi.org/10.9790/1684-1401012329

Bral, P., Tripathi, J. P., Dewangan, S., & Mahato, A. C. (2022). CFD analysis of an exhaust manifold for emission reduction. *Materials Today: Proceedings, 63*, 354-361. https://doi.org/10.1016/j.matpr.2022.03.202

De Angelis, G., & Palomba, F. (2004). *T*he Reliability Improvement of a Conventional Cast Iron Exhaust Manifold for a Small Size Gasoline Engine. *Internal Combustion Engine Division Fall Technical Conference*, 37467, 541-546. https://doi.org/10.1115/ICEF2004-0876

Deger, Y., Simperl, B., & Jimenez, L. P. (2004). Coupled CFD-FE-analysis for the exhaust manifold of a diesel engine. A*BAQUS Users' Conference*, 199-208. https://doi.org/10.1080/01430750.2017.1399457

Desai, A. R., Buradi, A., Gowthami, L., Praveena, B. A., Madhusudhan, A., & Bora, B. J. (2022). Computational Investigation of Engine Exhaust Manifold with Different Alternative Fuels By Using CFD. *2022 IEEE 2nd Mysore Sub Section International Conference (MysuruCon)*, 1-6. https://doi.org/10.1109/MysuruCon55714.2022.9972359

Deubert, D., Klingel, L., & Selig, A. (2024). Online simulation at machine level: a systematic review. *The International Journal of Advanced Manufacturing Technology, 131*(3), 977-998. https://doi.org/10.1007/s00170-024-13065-1

Guoquan, X., Huaming, W., Lin, C., & Xiaobin, H. (2021). Predicting unsteady heat transfer effect of vehicle thermal management system using steady velocity equivalent method. *Science Progress, 104*(2), 00368504211025933. https://doi.org/10.1177/00368504211025933

Haskew, H. M., & Liberty, T. F. (1991). In-use emissions with today's closed-loop systems. *SAE transactions*, 69-104. https://www.jstor.org/stable/44553584

Kresovic, U., Hussein, W., Zhou, C. Q., Majdak, J., & Cantwell, R. (2002). CFD Analysis of Liquid-Cooled Exhaust Manifolds in a Real Time Engine Cycle. *ASME International Mechanical Engineering Congress and Exposition*, 36355, 39-46. https://doi.org/10.1115/IMECE2002-32095

Kumar, R. R., Razak, A., Alshahrani, S., Sharma, A., Thakur, D., Shaik, S., Saleel, C. A., & Afzal, A. (2022). Vibration analysis of composite exhaust manifold for diesel engine using CFD. *Case Studies in Thermal Engineering, 32*, 101853. https://doi.org/10.1016/j.csite.2022.101853

Li, D. L., Yin, Y. F., Chen, G. F., Cui, C., & Han, B. (2012). Thermal fatigue analysis of the engine exhaust manifold. *Advanced Materials Research, 482*, 214-219. https://doi.org/10.4028/www.scientific.net/AMR.482-484.214

Lin, T. Y., Shi, G., Yang, C., Zhang, Y., Wang, J., Jia, Z., Guo, L., Xiao, Y., Wei, Z., & Lan, S. (2021). Efficient container virtualization-based digital twin simulation of smart industrial systems. *Journal of cleaner production, 281*, 124443. https://doi.org/10.1016/j.jclepro.2020.124443

Maheshappa, H., Pravin, V. K., Umesh, K. S., & Veena, P. H. (2013). Design analysis of catalytic converter to reduce particulate matter and achieve limited back pressure in diesel engine by CFD. *International Journal of Engineering Research and Applications (IJERA), 3*(1), 998-1004. https://doi.org/10.4271/2011-01-1245

Manohar, D. S., & Krishnaraj, J. (2018). Modeling and analysis of exhaust manifold using CFD. *IOP Conference Series: Materials Science and Engineering, 455*, 012132. https://doi.org/10.1088/1757-899X/455/1/012132

Prithvi, R. R., Midhun, M. P., & Prakash, D. (2020). Computational simulation and experimental validation of an engineering problem: A case study on heat transfer in cylindrical fin with phase‐change material. *Computer Applications in Engineering Education, 28*(1). https://doi.org/10.1002/cae.22183

Pathak, A., & Deshmukh, S. (2021). Thermal analysis of Exhaust manifold system using computational fluid dynamics. *Spectrum of Emerging Sciences, 1*(1), 50-55. https://esciencesspectrum.com/AbstractView.aspx?PID=2021-1-1-11

Rodríguez, B., González, F., Naya, M. Á., & Cuadrado, J. (2020). Assessment of methods for the real-time simulation of electronic and thermal circuits. *Energies, 13*(6), 1354. https://doi.org/10.3390/en13061354

Sadhasivam, C., Murugan, S., Vairamuthu, J., & Priyadharshini, S. M. (2021). Design and analysis of two-cylinder exhaust manifold with improved performance in CFD. *Materials Today: Proceedings, 37*, 2141-2144. https://doi.org/10.1016/j.matpr.2020.07.574

Sahoo, D. K., & Thiya, R. (2019). Coupled CFD–FE analysis for the exhaust manifold to reduce stress of a direct injection-diesel engine. *International Journal of Ambient Energy, 40*(4), 361-366. https://doi.org/10.1080/01430750.2017.1399457

Seenikannan, P., Periyasamy, V. M., & Nagaraj, P. (2008). An experimental analysis of a Y section exhaust manifold system with improved engine performance. *International Journal of Product Development, 6*(1), 50-56. https://doi.org/10.1504/IJPD.2008.019121

Sulistyo, B., Sofyan, H., Sukardi, T., & Widyianto, A. (2023). Performance and Emission

Characteristics Using Dual Injection System of Gasoline and Ethanol. *Automotive Experiences, 6*(2), 245-258. https://doi.org/10.31603/ae.8070

Taibani, A. Z., & Kalamkar, V. (2012). Experimental and computational analysis of behavior of three-way catalytic converter under axial and radial flow conditions. *International Journal of Fluid Machinery and Systems, 5*(3), 134-142. https://doi.org/10.5293/IJFMS.2012.5.3.134

Teja, M. A., Ayyappa, K., Katam, S., & Anusha, P. (2016). Analysis of exhaust manifold using computational fluid dynamics. *Fluid Mechanics Open Access 3*(1), 1000129. https://doi.org/10.4172/2476-2296.1000129

Umesh, K. S., & Rajagopal, V. P. K. (2013). CFD Analysis Of Exhaust Manifold Of Multi-Cylinder Si Engine Todetermine Optimal Geometry For Reducing Emissions. *International Journal of Automobile Engineering Research and Development*, 45-56. https://www.semanticscholar.org/paper/CFD-Analysis-of-Exhaust-Manifold-of-Multi-Cylinder-Umesh-Pravin/32377d5d689a52c4f7ecb1f9b7e27f313ac2a5de

Xu, S. L., Shi, Y., & Li, S. C. (2012). Heat Transfer and Thermal Load Analysis of Exhaust Manifold. *Applied Mechanics and Materials, 226*, 2240-2244. https://doi.org/10.4028/www.scientific.net/AMM.226-228.2240

Yogesh, K., Ananthesha, B., & Mahendra Babu, N. C. (2020). Assessment of thermo-mechanical fatigue performance of an exhaust manifold through simulation. *Indian Journal of Science and Technology, 10*(12), 1-6. https://doi.org/10.17485/ijst/2017/v10i12/105559