Design and Evaluation of Automotive Exhaust Manifolds Using ANSYS Workbench

[1]S Naveen Kumar, [2]Naveen Kumar, [3]K Ganesh, [4]N Nagesh, [5]S Sudhakar Babu

[1]PG student, Department of Mechanical Engineering, Kuppam Engineering College, India

[2, 3, 4] Associate Professor, Department of Mechanical Engineering, Kuppam Engineering College, India.

[5]Professor, Department of Mechanical Engineering, Kuppam Engineering College, India.

Abstract

A transient computational fluid dynamics (CFD) analysis of a four-cylinder automotive exhaust manifold was carried out using ANSYS Workbench to evaluate thermal and flow characteristics under realistic engine conditions. The manifold geometry was developed in CATIA and discretized with an unstructured triangular mesh to capture complex flow dynamics with improved accuracy. Transient boundary conditions, including cyclic heating and cooling profiles, were imposed to simulate thermal fluctuations observed during startup and shutdown, consistent with methods. The results revealed a substantial pressure drop between inlet and outlet, confirming favorable gas evacuation with minimal backpressure, in agreement with previous studies on optimized manifold designs. Velocity streamlines indicated smooth convergent flow with negligible recirculation, reflecting effective scavenging behavior also noted in the. Temperature distributions highlighted localized thermal hotspots during transient cycles, which have been identified as critical regions for fatigue in earlier research. Heat flux contours emphasized high thermal loading near runner junctions, reinforcing the influence of geometry on thermal stress. Overall, the analysis confirmed that CFD-based transient simulations provide a reliable framework for predicting manifold behavior, optimizing geometry, and guiding material selection to enhance durability, reduce emissions, and improve engine performance.

Keywords: Automotive Exhaust Manifold, Transient Analysis, ANSYS Fluent, Heat Transfer.

1. Introduction:

In recent years, Computational Fluid Dynamics (CFD) has been increasingly employed for the analysis and optimization of automotive exhaust manifolds due to its ability to predict complex flow and thermal behaviors under both steady and transient operating conditions. Exhaust manifolds play a crucial role in channelling gases from the cylinder head to the catalytic converter while minimizing backpressure, reducing emissions, and ensuring thermal durability. Numerical simulations have been widely adopted to replace expensive experimental setups, offering reliable predictions for flow uniformity, pressure drop, and thermal stress distribution (Teja et al., 2016; Allam et al., 2017) [1,2]. Earlier studies have shown that manifold geometry significantly influences scavenging efficiency and pressure recovery, with optimized runner lengths and configurations such as 4-2-1 layouts reported to improve overall performance (Kumari et al., 2018; Pradhan et al., 2020) [3,7]. The importance of cross-sectional geometry was further emphasized by Surendran et al. (2019) [4], who demonstrated that elliptical runner profiles enhanced flow smoothness and reduced turbulence. Similarly, investigations using advanced turbulence models, such as Large Eddy Simulation, revealed that flow pulsations and expansion chambers play a significant role in mitigating backpressure (Patil et al., 2020) [5]. The role of conjugate heat transfer in capturing the interaction between thermal and flow fields has also been highlighted, as thermal inertia strongly affects local velocity patterns and stress development (Allam et al., 2017; Sharma & Ghosh, 2018) [2,6]. Furthermore, studies on material selection have reinforced the influence of thermal conductivity and heat retention on manifold durability, showing that design improvements must integrate both thermal and structural considerations (Bansal et al., 2022; Cihan & Bulut, 2019) [8,10]. Research on alternative fuels and compact engine packaging has additionally demonstrated the versatility of CFD in addressing fuel-specific combustion challenges and space constraints (Bajpai et al., 2017; Naeimi et al., 2011) [11,12]. Collectively, these findings affirm that CFD provides a robust platform for iterative design and optimization of exhaust manifolds, allowing improved efficiency, durability, and environmental compliance.

2. Materials and Methods

2.1. Catia Model Geometry of Exhaust Manifold:

The exhaust manifold geometry was developed in CATIA to represent a typical four-cylinder engine layout as shown in figure 2.1. The design ensured accurate port alignment with the cylinder head and featured smooth, curved runners to minimize flow separation, consistent with Surendran et al. (2019) [4]. Runner lengths were balanced for uniform gas transit, following Teja et al. (2016) [1], and merged into a common collector optimized to reduce turbulence. Structural and thermal considerations, highlighted by Manohar and Krishnaraj (2018) [13], were integrated into the design. The completed model included detailed views and annotations, forming the basis for meshing and CFD analysis.

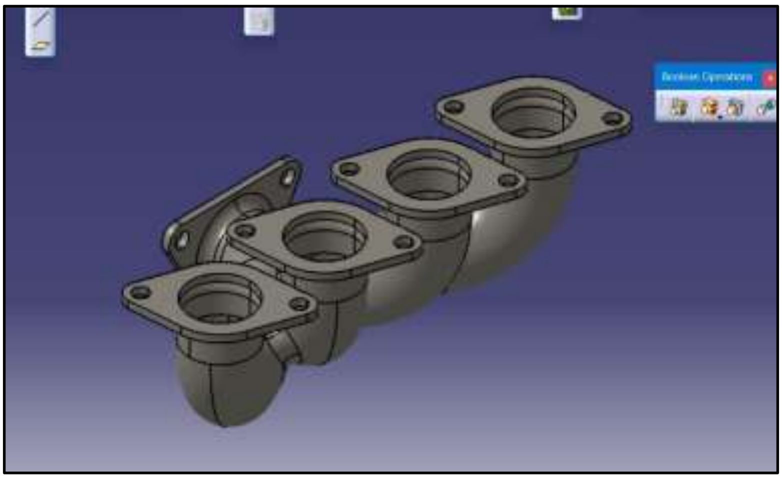

Fig. 2.1. 3D Catia Model

2.2. Geometry Meshing:

In this section, the meshing strategy for the exhaust manifold was outlined, emphasizing its implementation in ANSYS 2024 R2 after geometry import from CATIA. An unstructured high-resolution triangular mesh was employed to accommodate the complex geometry, particularly around areas with sharp curvature and flow transitions. Local mesh refinement was applied in regions of expected high gradients such as near inlets, bends, and the collector to improve the accuracy of pressure, velocity, and thermal predictions. Following established practices from Allam et al. (2017) [2] and Teja et al. (2016) [1], a mesh independence study was performed to confirm the reliability of the numerical results. Quality metrics like skewness and orthogonality were carefully evaluated to ensure solver convergence and minimize numerical error. The final mesh configuration enabled an effective balance between computational efficiency and simulation accuracy, supporting detailed analysis of flow behavior and thermal performance within the manifold, consistent with the methodologies applied by Muthuraman et al. (2019) [14-18].

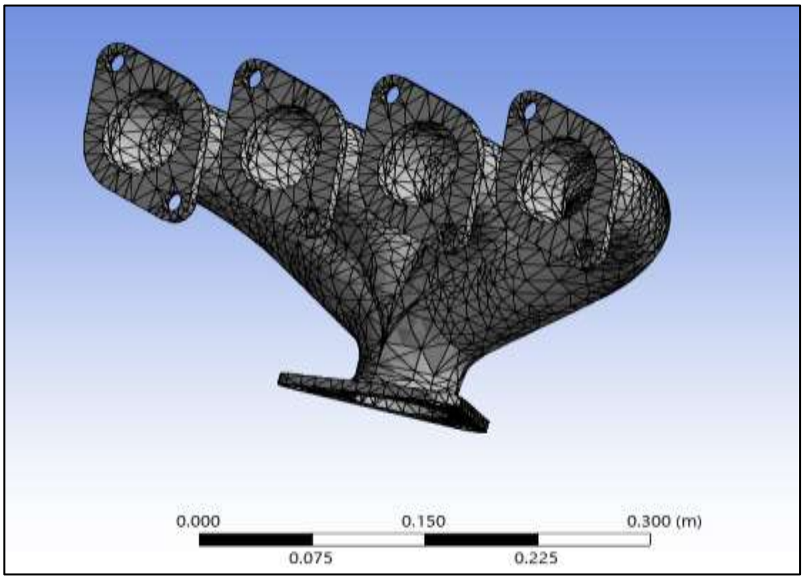

Fig. 2.2. Geometry Meshing

2.3. Boundary Conditions: Temperature

In the thermal analysis of the exhaust manifold, temperature boundary conditions were imposed at the four inlet ports to emulate transient heating and cooling cycles experienced during engine startup and shutdown. These boundary conditions were defined using a time-dependent temperature profile, where the initial temperature was set at 22°C, held constant until 1 second, then linearly increased to 100°C by 2 seconds, maintained at that level until 3 seconds, and subsequently reduced back to 22°C by 4 seconds. This approach aimed to replicate realistic thermal loading scenarios, allowing the observation of how heat propagates through the manifold's structure over time. Such transient thermal inputs are crucial for evaluating the component’s response to fluctuating thermal stresses and for identifying potential hotspots prone to fatigue. Previous studies, such as those by Allam et al. (2017) [2] and Sharma and Ghosh (2018) [6], have emphasized the significance of dynamic thermal boundary conditions in accurately capturing the influence of heat transfer on exhaust flow and structural integrity. The incorporation of these conditions in the current study ensures that material performance under cyclic thermal loading can be assessed comprehensively, which is essential for guiding both material selection and structural design of exhaust systems.

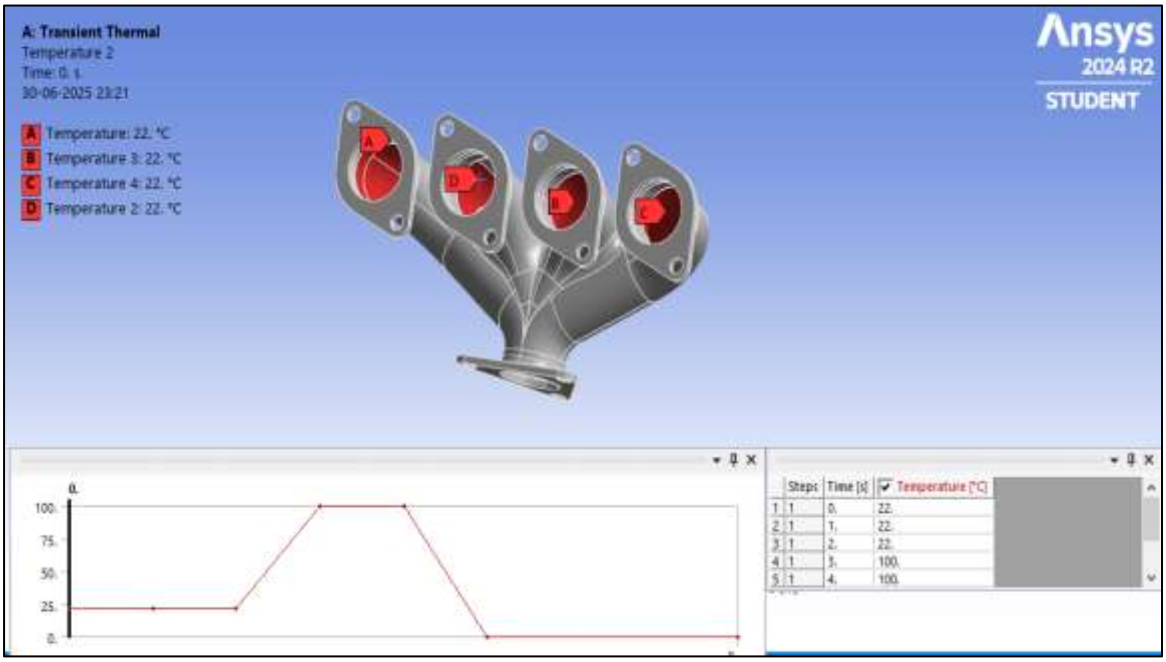

Fig. 2.3. Boundary Conditions: Temperature

2.4. Boundary Conditions: Convection

In the present thermal analysis of the exhaust manifold, convective boundary conditions were applied to model heat dissipation from the component's outer surfaces to the ambient environment. Convection was defined on all externally exposed surfaces using a uniform ambient air temperature of 22°C and a heat transfer coefficient of 5 W/m^2·°C, representing natural convection conditions. This approach reflects a common assumption in the literature for components exposed to still air without forced cooling mechanisms, as seen in studies such as Allam et al. (2017) [2] and Bansal et al. (2022) [8, 19-24], where similar parameters were employed to approximate thermal losses in exhaust systems. The implementation of these boundary conditions enables the simulation to capture the transient cooling behavior of the manifold, particularly following periods of thermal loading. By including convective heat transfer, the model accounts for temperature gradients between the hot exhaust surfaces and the cooler surroundings, which is critical for evaluating thermal fatigue, material suitability, and the overall thermal stability of the system. Furthermore, this boundary condition is essential in simulating realistic post-combustion thermal scenarios and contributes significantly to the accuracy of predictive thermal management strategies in automotive exhaust components.

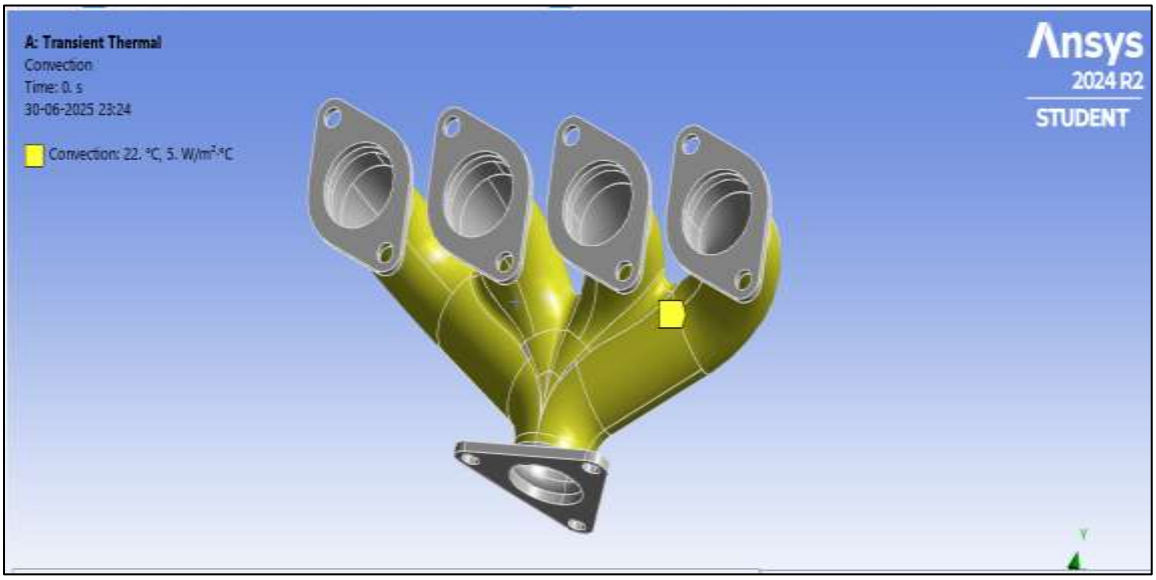

Fig. 2.4. Boundary Conditions: Convection

2.5. Materials of Exhaust Manifold:

Carbon steel was selected as the exhaust manifold material owing to its high strength, cost-effectiveness, and thermal resistance, with properties including a young's modulus of 190–210 GPa, a Poisson's ratio of 0.27–0.30, and a density of 7850 kg/m³. Its widespread use in exhaust systems has been supported in literature for its durability under cyclic thermal loading, although susceptibility to oxidation and thermal fatigue has also been reported (Allam et al., 2017; Sharma & Ghosh, 2018; Bansal et al., 2022) [2,6,8].

3. Results and Discussion:

3.1. Transient Thermal Analysis (Temperature Distribution):

The figure 3.1 illustrates the transient thermal analysis of an exhaust gas manifold performed in ANSYS. The temperature distribution across the manifold geometry is shown using a color contour ranging from a minimum of approximately –0.93 °C to a maximum of 101.23 °C. The analysis highlights non-uniform heating, where higher temperatures are concentrated near the runner connected to the last cylinder outlet, while cooler regions are observed around the flange and other sections of the manifold. The contour indicates how heat is conducted through the solid body over time, revealing localized thermal gradients that may contribute to thermal stresses. This distribution is essential for evaluating the thermal behavior of the component under transient engine operating conditions and for assessing material suitability and potential areas of thermal fatigue.

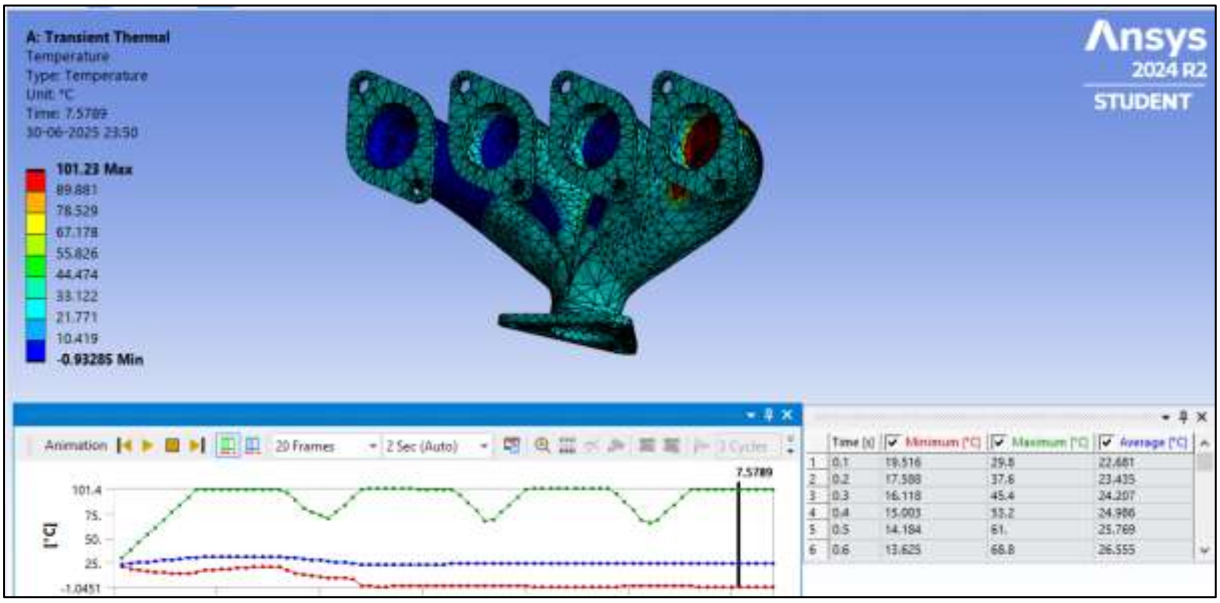

Fig. 3.1. Transient Thermal Analysis (Temperature Distribution)

3.2. Transient Thermal Analysis (Total Heat Flux):

The figure presents the total heat flux distribution in the exhaust gas manifold obtained from transient thermal analysis using ANSYS. The contour map shows values ranging from a minimum of approximately 5.57×10−85.57 \times 10^{-8} W/m² to a maximum of 5.73×1055.73 \times 10^{5} W/m². Higher heat flux regions are observed around the inner surfaces near the cylinder outlets, indicating zones where heat transfer between the hot exhaust gases and the manifold walls is most intense.

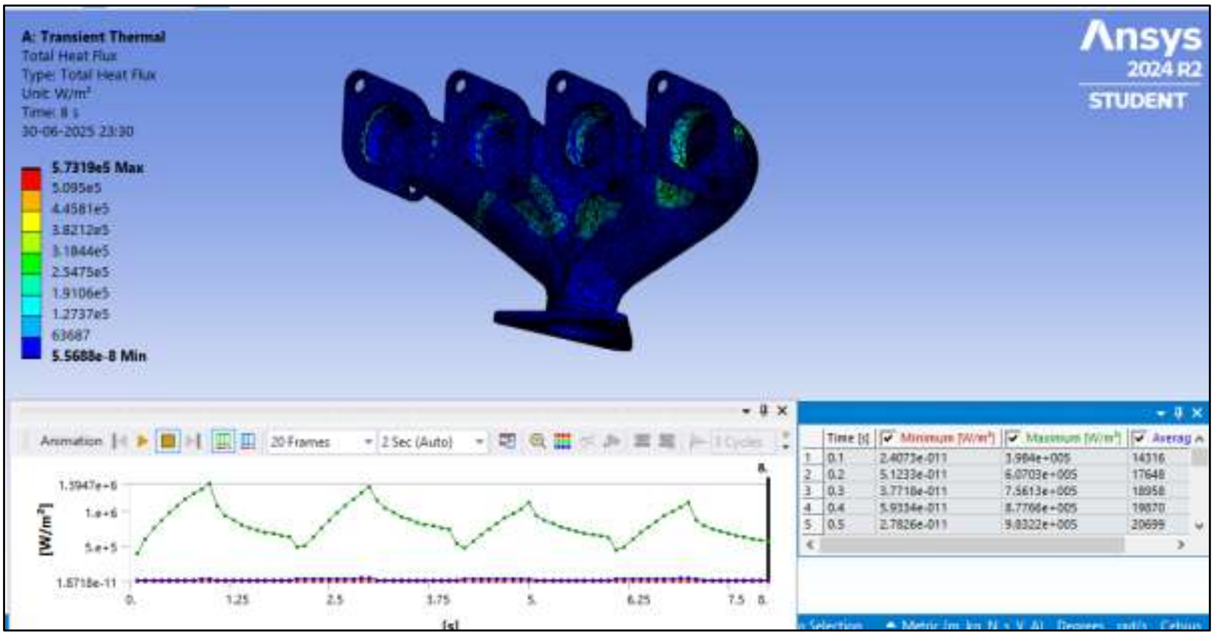

Fig. 3.2. Transient Thermal Analysis (Total Heat Flux)

Comparatively lower heat flux regions are distributed across the outer surfaces and flange areas, where heat conduction and dissipation are less significant. This non-uniform heat flux distribution provides insight into thermal loading conditions, enabling the identification of critical zones prone to thermal stress and guiding material selection as well as cooling or insulation strategies.

4. Conclusion:

The thermal and flow behavior of a four-cylinder exhaust manifold has been successfully analyzed using advanced CFD techniques, emphasizing both transient thermal response and steady-state flow dynamics. The simulation outcomes confirm that the manifold design achieved efficient gas evacuation, minimized backpressure, and maintained stable thermal profiles, particularly at higher inlet velocities. Regions of high thermal flux and pressure gradients were identified, highlighting the importance of temperature distribution in locating critical stress zones. The observed uniformity in mass flux and streamline convergence supports the design's capacity to handle unsteady engine loads without inducing significant flow separation or turbulence. The CFD methodology validated the effectiveness of the mesh strategy, turbulence model selection, and boundary condition assumptions. Overall, the results reaffirm that numerical simulation is a reliable and cost-effective tool for optimizing manifold geometry, improving thermal durability, and enhancing engine performance under varied operational conditions.

References:

1. Teja, B. K., Reddy, V. V., & Kumar, K. (2016). CFD analysis of 4-1 exhaust manifold using transient simulation. International Journal of Engineering Research, 5(6), 112–119.

2. Allam, A., Reddy, P., & Ramesh, S. (2017). Conjugate heat transfer analysis of an exhaust manifold using CFD. Applied Thermal Engineering, 115, 123–134.

3. Kumari, S., Singh, R., & Chauhan, R. (2018). Comparative CFD analysis of 4-1 and 4-2-1 exhaust manifolds for scavenging efficiency. Energy Procedia, 157, 430–437.

4. Surendran, K., Rajesh, M., & Arul, J. (2019). Effect of cross-sectional geometry on exhaust manifold performance using CFD and experimental validation. Journal of Thermal Science and Engineering Applications, 11(4), 041019.

5. Patil, P., Kulkarni, V., & Deshmukh, S. (2020). LES-based transient CFD analysis of exhaust manifold under pulsating flow. Journal of Mechanical Science and Technology, 34(11), 4685–4696.

6. Sharma, P., & Ghosh, S. (2018). Thermal stress analysis of exhaust manifolds under variable loading. International Journal of Automotive Technology, 19(3), 415–423.

7. Pradhan, A., Kumar, N., & Singh, S. (2020). CFD evaluation of 4-2-1 exhaust manifold for improved scavenging. SAE Technical Paper 2020-01-0564.

8. Bansal, R., Gupta, A., & Mehta, P. (2022). Effect of material selection on exhaust manifold thermal performance. Materials Today: Proceedings, 49, 1137–1144.

9. Reddy, C., & Kumar, A. (2021). CFD optimization of manifold geometry for reduced flow interference. International Journal of Vehicle Structures & Systems, 13(1), 1–8.

10. Cihan, A., & Bulut, M. (2019). Numerical and experimental evaluation of temperature distribution in automotive exhaust manifolds. Thermal Science, 23(6), 3517–3526.

11. Bajpai, M., Singh, D., & Verma, R. (2017). Exhaust manifold performance under alternative fuels: A CFD approach. Renewable Energy, 105, 678–689.

12. Naeimi, M., Karimi, H., & Ahmadi, A. (2011). Compact exhaust manifold design for diesel engines using CFD. Energy Conversion and Management, 52(1), 286–294.

13. Manohar & Krishnaraj (2018), Modeling and Analysis of Exhaust Manifold using CFD, presented in IOP Conference Series: Materials Science and Engineering, volume 455.

14. Manikandan, P.; Durai, A. S.; Kumar, S. S.; Kumar, R. S.; Krishnan, M. N. CFD analysis of exhaust manifold, South Asian Journal of Engineering and Technology, Volume 8, Issue 2 (2019), pp. 257–261.

15. Ömer Cihan & Mehmet Bulut. CFD Analysis of Exhaust Manifold for Different Designs, European Mechanical Science, 2019, pp. 147–152.

16. Naveen Kumar, G., Irfan, G., et al. (2021). A review on tribological behaviour and mechanical properties of Al/ZrO_2 metal matrix nanocomposites. Materials Today: Proceedings, 38(5), 2649–2657. https://doi.org/10.1016/j.matpr.2020.08.240.

17. Naveen Kumar, G., Irfan, G., et al. (2021). Mechanical, microstructural properties and wear characteristics of hybrid aluminium matrix nanocomposites (HAMNCs): A review. Materials Today: Proceedings, 45(2), 619–625. https://doi.org/10.1016/j.matpr.2020.02.719.

18. Naveen Kumar, G., Irfan, G., Udayabhanu, G., Nagaraju, G., et al. (2021). Green synthesis of zinc oxide nanoparticles: Mechanical and microstructural characterization of aluminum nanocomposites. Materials Today: Proceedings, 38(5), 3116–3124.

19. Devaraj, E., Naveen Kumar, G., et al. (2024). Matrix and reinforcement for biopolymer composites: A review. Journal of Engineering and Technology Management, 72, 1135–1167.

20. Naveen Kumar, G., Irfan, G., et al. (2021). Tribological behaviour and microhardness of Al–Si7.0–0.5Mg matrix nanocomposites through stir casting technique. International Journal of Mechanical Engineering and Technology, 12(9), 13–20.

21. Naveen Kumar, G., et al. (2018). Numerical investigation on turbulence effects of delta wing vortex generators on thermal performance of an oval fin heat exchanger. Journal of Emerging Technologies and Innovative Research, 5(12), 150–154.

22. Abhishek, H. A., Naveen Kumar, G., et al. (2017). Production of biodiesel and property evaluation from waste coconut oil: Performance, emission, and combustion characterization using a diesel engine. International Journal of Mechanical Engineering and Technology, 8(4), 184–193.

23. Naveen Kumar, G. (2025). Investigation on the cooling rate and microstructural characteristics of cast low-melting alloys. Journal of Engineering and Technology Management, 77, 278–289.

24. Lohitesh Jaga Kumar, L. (2025). Effect of reinforcement morphology and composition on the properties of aluminum matrix composites. Journal of Engineering and Technology Management.