|| Volume 10|| Issue 2 || 2026 || ISSN (Online) 2456-0774 INTERNATIONAL
JOURNAL OF ADVANCE SCIENTIFIC RESEARCH
AND ENGINEERING TRENDS

Flow Analysis of Car AC Duct to Find Temperature, Velocity and Pressure Difference

Girdhar Shendre[1], Sarvadnya Vilas Ingole[2], Vaidehi Sudhir Gorale[3], Sarang Kalyanrao Behare[4], Aman Sunil Gayakwad[5], Arpit Sanjay Gadwe[6], Ananad Ajay Shende[7]

DRGITR AMRAVATI

***

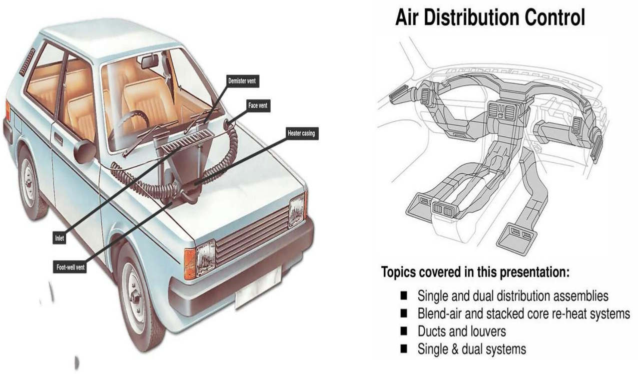
Abstract: Airflow management represents a critical and multidisciplinary aspect of modern automotive engineering, influencing passenger thermal comfort, system efficiency, aerodynamic drag, and electric vehicle driving range. As the global automotive industry continues its transition toward electrification and sustainability, optimizing both internal heating, ventilation, and air-conditioning (HVAC) airflow systems and external aerodynamic characteristics has become increasingly essential. This research presents a comprehensive Computational Fluid Dynamics (CFD)-based investigation integrating internal duct flow optimization and external aerodynamic drag reduction through S-duct implementation. Reynolds-Averaged Navier–Stokes (RANS) simulations were employed to analyze airflow behavior under steady-state conditions. Internal HVAC duct modifications, including outlet geometry transformation and elbow angle optimization, resulted in airflow velocity improvements ranging from 4% to 9% while maintaining outlet velocity uniformity within 1.3%. External aerodynamic optimization using an S-duct configuration demonstrated a measurable reduction in drag coefficient, corresponding to an estimated 48 km improvement in electric vehicle driving range under highway conditions. The results highlight the importance of adopting a holistic airflow optimization strategy that integrates internal thermal management and external aerodynamic performance to improve comfort, efficiency, and sustainability simultaneously.

Keywords: *CFD Tool, AC Duct*

***

I.INTRODUCTION:

In contemporary automotive design, airflow optimization has evolved beyond a simple comfort requirement into a comprehensive engineering challenge involving thermal management, aerodynamic performance, energy efficiency, and environmental sustainability. The heating, ventilation, and air-conditioning system plays a vital role in maintaining thermal comfort within the passenger cabin by regulating air temperature, humidity, and distribution. However, packaging constraints within the engine compartment and dashboard structure impose geometric limitations that often result in complex duct configurations. These geometries frequently include sharp bends, sudden cross-sectional transitions, and irregular outlet shapes, which generate turbulence, pressure losses, and uneven airflow distribution. Such inefficiencies compromise passenger comfort and increase the energy demand of the HVAC system.



Topics covered in this presentation:

- Single and dual distribution assemblies
- Blend-air and stacked core re-heat systems
- Ducts and louvers
- Single & dual systems

Fig. 1: Car Air Conditioning System

The significance of internal airflow optimization becomes even more pronounced in electric vehicles, where HVAC energy consumption directly affects battery discharge rates and overall lithium-ion batteries, while offering high energy density (~150- driving range. Simultaneously, external aerodynamic performance significantly influences total vehicle efficiency. At higher speeds, aerodynamic drag becomes the dominant resistive force acting on the vehicle, and the power required to overcome drag increases proportionally to the square of velocity. Consequently, even minor improvements in drag coefficient can produce substantial gains in energy efficiency and driving range.Traditionally, internal airflow design and external aerodynamic optimization have been treated as independent engineering domains. However, both systems directly influence total vehicle energy consumption and must be optimized in an integrated manner. This study presents a unified CFD-based approach to optimize internal HVAC duct geometry and external vehicle aerodynamics simultaneously.

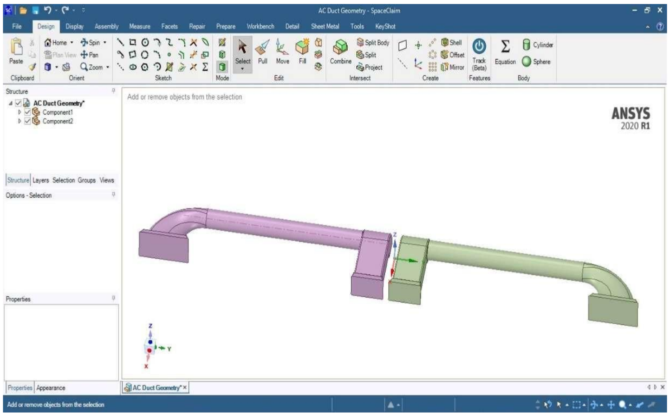
Theoretical Background

Airflow within automotive HVAC ducts is characterized by turbulent behavior due to high blower speeds and geometric complexity. The governing principles of fluid motion are described by the conservation equations of mass, momentum, and energy. The continuity equation ensures mass conservation, while the Navier–Stokes equations describe momentum transfer under viscous and pressure forces. In practical CFD simulations, these equations are solved in their Reynolds-averaged form to account for turbulence effects.

Turbulence modeling is essential because direct numerical simulation of turbulent automotive flows is computationally impractical. Several turbulence models were evaluated in this research, including the standard k–ε, realizable k–ε, RNG k–ε, standard k– ω, and SST k–ω models. The SST k–ω model was selected due to its superior ability to capture boundary layer behavior, adverse pressure gradients, and flow separation, making it suitable for both internal duct flow and external aerodynamic simulations.Aerodynamic drag acting on a vehicle consists primarily of pressure drag and skin friction drag. Pressure drag arises from flow separation and wake formation behind the vehicle, while skin friction drag results from viscous shear stresses along the surface. The drag force is proportional to air density, frontal area, drag coefficient, and the square of vehicle velocity. Therefore, minimizing drag coefficient through optimized airflow management is critical for improving vehicle efficiency.

II.METHODLOGY

The research methodology involved detailed three-dimensional CFD simulations under steady-state conditions. Internal HVAC duct geometry was developed using CAD modeling, incorporating baseline rectangular outlet shapes and a 75-degree elbow configuration. Unstructured tetrahedral meshes were generated, and boundary layer refinement was applied near walls to accurately capture shear stresses and velocity gradients. A grid independence study was conducted to ensure that simulation results were not influenced by mesh density. Convergence criteria were established based on residual reduction and stabilization of monitored parameters such as outlet velocity and drag coefficient.For internal airflow analysis, mass flow inlet boundary conditions were applied at the blower entrance, while pressure outlet conditions were prescribed at each ventilation outlet. No-slip wall conditions were imposed on all duct surfaces. Parametric modifications included transforming outlet shapes from rectangular to circular configurations and reducing elbow angles from 75 degrees to 65 degrees to examine their impact on velocity distribution and pressure losses.



External aerodynamic simulations were performed on a full vehicle geometry model under highway velocity conditions. Symmetry boundary conditions were utilized to reduce computational cost without compromising solution accuracy. The S-duct was integrated into the vehicle front section to channel high-pressure air from the bumper region toward the hood surface. Drag coefficient values were calculated for baseline and modified configurations to quantify aerodynamic improvements.

Results and Discussion

The baseline HVAC duct configuration exhibited moderate pressure losses and minor but noticeable velocity imbalance among outlets. Flow visualization revealed secondary vortices and turbulence formation near elbow sections, which contributed to energy losses and reduced flow efficiency. Upon converting rectangular outlets into circular geometries, hydraulic resistance decreased due to improved perimeter-to-area ratio, resulting in airflow velocity improvements between 6% and 8%. The circular design reduced boundary layer separation and minimized localized turbulence.Further optimization was achieved by reducing the elbow angle from 75 degrees to 65 degrees. This modification improved flow smoothness and reduced adverse pressure gradients. Combined geometric improvements resulted in total airflow velocity enhancement ranging from 4% to 9% compared to the baseline case. Additionally, outlet velocity variation was reduced to approximately 1.3%, ensuring uniform air distributiona cross the cabin. Such uniformity is essential for achieving consistent thermal comfort for all passengers and reducing blower workload, which in turn lowers energy consumption.In the aerodynamic analysis, the baseline vehicle configuration demonstrated significant stagnation pressure at the front bumper and flow separation over the hood and windshield region. These phenomena contributed to increased pressure drag and enlarged wake formation behind the vehicle. Implementation of the S-duct effectively redirected high- pressure airflow from the bumper region to the hood surface, reducing stagnation pressure and improving boundary layer attachment. The optimized S-duct configuration demonstrated the most substantial drag coefficient reduction among the tested designs.Energy analysis indicated that the aerodynamic improvement could extend electric vehicle driving range by approximately 48 kilometers under comparable operating conditions. This enhancement results from reduced propulsion power demand required to overcome aerodynamic resistance. The combined effect of internal HVAC optimization and external aerodynamic improvement leads to significant total energy savings, particularly for electric vehicles where both systems directly influence battery discharge rates.

Integrated Engineering Implications

The integration of internal and external airflow optimization provides synergistic performance benefits. Internal duct modifications enhance passenger comfort and reduce HVAC energy demand, while aerodynamic improvements reduce propulsion energy losses at higher speeds. When implemented simultaneously, these improvements contribute to lower overall energy consumption, improved electric vehicle range, reduced operational cost, and decreased environmental impact.From a manufacturing perspective, CFD-based optimization significantly reduces development time and cost by minimizing reliance on physical prototyping. Virtual simulations allow rapid evaluation of multiple geometric configurations, enabling efficient design iteration and performance validation. This approach aligns with modern digital engineering practices and supports sustainable vehicle development goals.

Validation and Limitations

The selected turbulence model was validated against benchmark cases and available literature data to ensure predictive reliability. Drag coefficient predictions showed acceptable deviation within engineering tolerance limits. However, certain limitations remain in the present study. The simulations were conducted under steady-state conditions and did not include transient thermal coupling with cabin occupants. Additionally, experimental wind tunnel validation was not performed within the scope of this work. Future investigations should incorporate transient simulations, experimental verification, and real-world driving cycle analysis to further validate and refine the findings.

III.CONCLUSION

This comprehensive investigation demonstrates that integrated CFD-based optimization of automotive airflow systems significantly enhances thermal comfort, energy efficiency, and aerodynamic performance. Internal HVAC duct modifications, including circular outlet transformation and elbow angle optimization, improved airflow velocity by up to 9% while maintaining high uniformity. External S-duct implementation reduced aerodynamic drag and increased electric vehicle driving range by an estimated 48 kilometers. The results confirm that airflow optimization should be approached holistically rather than as isolated internal and external problems. By combining thermal management improvements with aerodynamic enhancements, automotive manufacturers can achieve measurable performance gains that contribute to sustainability, efficiency, and passenger comfort.

IV.REFERENCES

[1] Libin Tan, Yuejin Yuan, “Computational fluid dynamics simulation and performance optimization of an electrical vehicle Air-conditioning system”. Received 20 July 2020; revised 18 April 2021; accepted 1 May 2021, Available online 5 June 2021, 1110-0168 _ 2021 THE AUTHORS. Published by Elsevier BV on behalf of Faculty of Engineering, Alexandria University. This is an open access article under the CC BY- NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/)

[2] Ashwanee Patel, “CFD Analysis of Air Conditioning in Room Using Ansys Fluent”. Nagaji Institute of Technology and Management, Thakur Baba Campus, Jhansi Road, Sitholi, Gwalior -474001

[3] Jiying Liu 1,2,* , Shengwei Zhu 2, Moon Keun Kim 3,* and Jelena Srebric 2, “A Review of CFD Analysis Methods for Personalized Ventilation (PV) in Indoor Built Environments”. Sustainability 2019, 11, 4166; doi:10.3390/su11154166 www.mdpi.com/journal/sustainability

[4] K.M. Swaminadh 2k. Bicha, “CFD analysis on mass fraction of refrigerant leak and temperature through automobile HVAC ducts”. Vol 10, Issue 12, Dec / 2019, ISSN NO: 0377-9254, www.jespublication.com

[5] Haribabu Krishnaswamy A*, Sivaprakash Muthukrishnan B, Sathish Thanikodi C, Godwin Antony Arockiaraj D, And Vijayan Venkatraman, “Investigation Of Air Conditioning Temperature Variation By Modifying The Structure Of Passenger Car Using Computational Fluid Dynamics”. Thermal Science, Year 2020, Vol. 24. No. B1, PP 495-498,

[6] Senveli, T. Dizman, A. Celen, “CFD analysis of smoke and temperature control system of an indoor parking lot with jet fans”. Journal of Thermal Engineering http://eds.yildiz.edu.tr/journal-of-thermal-engineering/Articles, Yildiz Technical University Press, Istanbul, Turkey Manuscript Received August 28, 2014; Accepted September 05, 2014, Vol. 1, No. 2, pp. 116-130, April, 2015.

[7] Rameshkumar.A1, Jayabal.S2, and Thirumal.P3., “CFD Analysis of Air Flow and Temperature Distribution in an Air Conditioned Car”. International Refereed Journal of Engineering and Science (IRJES) ISSN (Online) 2319-183X, (Print) 2319-1821 Volume 2, Issue 4(April 2013), PP.01-06 www.irjes.com

[8] U. V. Narayana Rao, K. Srinivasa Rao, A. Dattatreya Kumar, G. Yaswanth Kumar, G. Sridhar Babu, “CFD Analysis of Auditorium using Ansys Fluent”. International Journal of Recent Technology and Engineering (IJRTE) ISSN: 2277-3878 (Online),Volume-8 Issue-2, July 2019

[9] 1Megha Shukla, 2Lakshminarasimha N, “CFD Analysis of Airflow inside a Car Compartment”. International Journal of Technical Innovation in Modern Engineering C Science (IJTIMES) Impact Factor: 5.22 (SJIF-2017), e-ISSN: 2455-2585 Volume 4, Issue 6, June-2018

[10] Shamim Ibrahim1, Arun S Lal2, RC Metha3, “CFD Investigation of Cabin Air Flow and Finding Out the Most Efficient Duct Combinations”. IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321- 7308