IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

Optimization In Hvac Duct Systems Of Vehicle Using Computational Fluid Dynamics (Cfd) Analysis.

Prashant Mandlik, M.Tech (Heat Power Engineering)

G. H. Raisoni College of Engineering & Management, Pune, India

Dr. Anil Sahu (Professor)
Department of Mechanical Engineering,
G. H. Raisoni College of Engineering & Management, Pune, India

Abstract: The growing demand for enhanced thermal comfort, energy efficiency, and compact design in modern vehicles necessitates the optimization of Heating, Ventilation, and Air Conditioning (HVAC) systems. HVAC duct design plays a pivotal role in determining airflow distribution, noise levels, and overall system efficiency. This project focuses on the optimization of HVAC duct systems in passenger vehicles through the application of Computational Fluid Dynamics (CFD) analysis. The primary objective is to reduce pressure losses, improve airflow uniformity, and enhance thermal comfort without compromising the packaging constraints of the vehicle interior.

The study begins with a baseline evaluation of a conventional HVAC duct design using CFD simulations to identify inefficiencies such as turbulence zones, high-pressure drops, and flow separation areas. Using this baseline as a reference, various design modifications are proposed, including changes in duct geometry, bends, splitter locations, and vent positions. Each modified design is analysed through CFD tools to assess parameters such as velocity distribution, static pressure, turbulence intensity, and flow uniformity.

Optimization techniques, including parametric analysis and design of experiments (DOE), are employed to systematically refine the duct design. The final optimized design demonstrates a significant reduction in pressure drop (by 6 pascal), improved flow uniformity at passenger outlets, and enhanced overall HVAC performance. Additionally, noise generation due to turbulent flow is minimized, contributing to improved cabin comfort.

The project validates the effectiveness of CFD as a powerful tool in the automotive HVAC system design process. The results highlight the potential for computational analysis to accelerate development cycles, reduce physical prototyping costs, and deliver better-performing systems. This study serves as a framework for future research and development of energy-efficient and ergonomically optimized vehicle HVAC systems.

***Index Terms* -** CFD, Ansys Fluent, HVAC Duct.

1.INTRODUCTION

1.1 Background and Significance

In today's automotive industry, ensuring passenger comfort and energy efficiency is a top priority. One of the most critical subsystems responsible for interior climate control is the Heating, Ventilation, and Air Conditioning (HVAC) system. HVAC systems regulate temperature, humidity, and air quality within the cabin, directly impacting occupant comfort, safety, and overall vehicle experience. As consumer expectations grow and regulatory requirements become stricter particularly in terms of fuel efficiency and emissions automakers are increasingly focusing on optimizing HVAC performance without increasing energy consumption.

A key component of any automotive HVAC system is its ductwork. The duct system is responsible for transporting conditioned air from the HVAC unit to various zones in the passenger cabin. It must do this efficiently, quietly, and uniformly, all within the tight spatial and structural constraints of a vehicle's interior. Poor duct design can lead to several issues, such as uneven air distribution, excessive pressure losses, increased noise (due to turbulence), and higher energy consumption as the blower motor compensates for inefficiencies.

1.2 Motivation for the Study

Conventional duct design methodologies rarely heavily on empirical rules, physical prototyping, and iterative testing, which can be time-consuming, expensive, and often sub-optimal. With the growing complexity of vehicle interiors—particularly in electric vehicles (EVs), which have different thermal management requirements—there is a pressing need for more advanced, data-driven methods of HVAC duct optimization. Computational Fluid Dynamics (CFD) presents a robust solution to this challenge. CFD enables engineers to visualize, analyse, and optimize airflow behaviour within the HVAC duct network using numerical simulations. It provides detailed insights into fluid flow characteristics, such as velocity profiles, turbulence intensity, pressure drops, and temperature gradients, which are otherwise difficult or impossible to measure accurately in physical tests. By integrating CFD early in the design cycle, manufacturers can accelerate development, reduce reliance on physical prototypes, and achieve better overall system performance.

This project leverages CFD analysis to optimize the HVAC duct system in a vehicle, with the goal of improving airflow efficiency, reducing pressure losses, minimizing noise, and ensuring uniform air distribution. The study explores various duct geometries and configurations to identify and implement an optimal design.

1.3 Problem Statement

The primary objective of this project is to address inefficiencies in the existing HVAC duct design in a typical passenger vehicle. Common issues such as flow separation, high-pressure loss, and uneven airflow distribution lead to reduced thermal comfort and increased energy consumption. These issues are often rooted in suboptimal duct geometry, poor layout of bends and splits, and inconsistent outlet performance.

Specifically, the problems include:

- High-pressure drop across the duct system, increasing the blower workload.
- Non-uniform air distribution at various passenger vents, leading to inconsistent cabin comfort.
- Unwanted noise caused by turbulence and sudden flow direction changes.
- Space constraints and packaging limitations impacting duct routing.

By applying CFD simulations and iterative design optimization, the project aims to propose an improved duct configuration that overcomes these challenges while remaining manufacturable and practical.

1.4 Objectives of the Study

The key objectives of this project are:

- To analyse the baseline HVAC duct system using CFD simulations to understand flow characteristics and identify performance bottlenecks.
- To explore alternative duct geometries and configurations through design modifications.
- To perform CFD simulations on modified designs and evaluate performance based on criteria such as pressure drop, flow uniformity, and turbulence levels.
- To implement an optimization strategy (such as parametric variation or Design of Experiments) for achieving the most efficient design.
- To validate the final optimized design by comparing its performance with the baseline in terms of airflow efficiency and passenger comfort.

1.5 Scope of the Study

This study focuses solely on the internal airflow characteristics of the HVAC duct system within the vehicle cabin. It does not include the thermal performance of the HVAC components such as the evaporator, heater core, or refrigerant system. Similarly, the study does not cover external airflow over the vehicle or interactions with under hood thermal management systems.

The analysis is confined to:

- The air distribution ducting network connecting the HVAC unit to passenger vents.
- Key components such as bends, T-junctions, diffusers, and outlets.
- Simulations under steady-state operating conditions.
- CFD simulations using a finite volume-based commercial solver (e.g., ANSYS Fluent, STAR-CCM+).

The outcome of the study will be an optimized duct design proposal supported by CFD results, including visualizations of airflow, pressure contours, and performance metrics.

2. LITERATURE REVIEW

2.1 Impact of realistic boundary conditions on CFD simulations: A case study of vehicle ventilation.

Florin Bode, Costin Coșoiu, Titus Joldos , Gabriel Mihai Sirbu, Paul Danca and Ilinca Nastase, have carried out the study on impact of realistic boundary condition on CFD simulation in vehicle ventilation. the study highlights the importance of integrating 3D HVAC ducts into personalized ventilation (PV) numerical simulations.

(Article Published in https://doi.org/10.1016/j.buildenv.2024.112264)

2.2 Computational fluid dynamics simulation and Performance optimization of an electrical vehicle Air-conditioning system.

Libin Tan and **Yuejin Yuan** have studied by considering full cabin along with HVAC unit done the CFD simulation and optimization of EV Air-conditioning system

(Article Published in https://doi.org/10.1016/j.aej.2021.05.001*)*

2.3 3-D NUMERICAL AND EXPERIMENTAL ANALYSIS FOR AIRFLOW WITHIN A PASSENGER COMPARTMENT

C.H. Chien, J.Y. Jang, Y.H. Chen and S.C. WU have done the air flow distribution analysis using thermotical and numerical method. Their study focused on comparison of thermotical and numerical method.

Published in International Journal of Automotive Technology, Vol. 9, No. 4, pp. 437−445 (2008)

3. METHODOLOGY

3.1 Geometry Information

In this study input CAD received and model preparation has been done in Software shown in Figure 3.1.

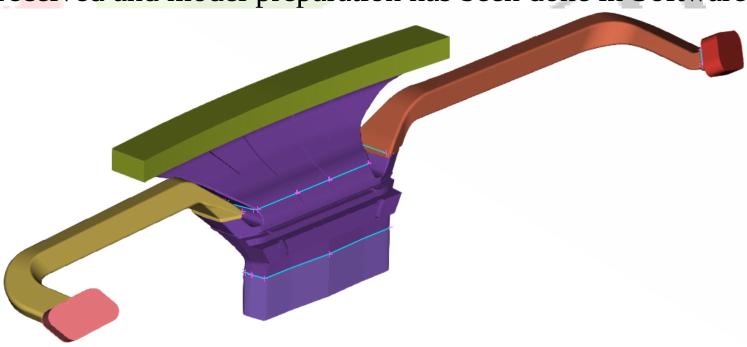

Fig no.1 Final model

3.2 Meshing and Quality Analysis

The geometry of the HVAC duct system was discretized using a surface mesh consisting of approximately 80,000 elements. The meshing process was performed to balance between capturing flow details around critical features (e.g., bends, branches, and outlets) and maintaining reasonable computational cost.

In this study, the average skewness was maintained at 0.6, which falls within the acceptable quality range. This indicates that while the mesh may not be optimal in all regions, it is sufficiently robust for preliminary simulations and optimization studies. Special attention was given to areas with complex geometry (e.g., sharp turns and splitters), where local refinement was applied to improve resolution and minimize numerical diffusion.

3.3 Volume meshing using ANSYS Fluent

Volume meshing was carried out in ANSYS Fluent using tetrahedral elements, with a total mesh count of approximately **0.9 million elements**. The mesh was generated with a **growth rate of 1.1** and a **maximum cell length of 12 mm** to ensure smooth transition and adequate resolution across the duct geometry. Upon quality assessment, the mesh exhibited a **maximum skewness of 0.9**, which falls in the upper acceptable limit for CFD simulations, though further refinement may be required for improved numerical accuracy.

3.4 CASE Setup Using ANSYS Fluent

3.4.1 General setting

The geometry model was scaled and converted into meters to ensure consistency in units. A pressure-based solver was selected for the simulation, with absolute velocity formulation and a steady-state time setting to analyze the airflow behavior under stable operating conditions.

3.4.2 Selection of model

The k-ε turbulence model was selected for the simulation, specifically using the Realizable k-ε formulation in combination with the Standard Wall Function approach to accurately capture near-wall flow behavior and turbulence effects within the HVAC duct system

3.4.3 Selection of material

The duct material was defined as aluminum, with a density of 2719 kg/m³, to represent the structural properties of the HVAC system. The working fluid was specified as air, with a density of 1.215 kg/m³, to simulate realistic airflow conditions within the duct.

3.4.4 Boundary condition

In the simulation, a mass flow inlet boundary condition was applied with a mass flow rate of 0.16122 kg/s to represent the incoming airflow, while the outlet was defined as a pressure outlet, allowing the air to exit the system at atmospheric pressure.

3.4.5 Selection of method and controls

For the simulation, the pressure-velocity coupling was handled using the Coupled scheme to ensure strong coupling between pressure and velocity fields. The Pressure discretization was set to Second Order, while both the Momentum and Turbulent Kinetic Energy equations were solved using the Second Order Upwind scheme to improve solution accuracy. A flow Courant number of 70 was used to control numerical stability and convergence behavior.

3.4.6 Report definition

Report definitions were configured primarily for reference purposes; however, only the key results of interest were selected for monitoring during the simulation to streamline the solution process and focus on critical performance parameters.

4. RESULTS AND DISCUSSION

4.1 Velocity flow path distribution on base case and optimize case.

The velocity flow path within the duct was visualized to analyze airflow behavior, revealing the distribution and direction of air throughout the system, and highlighting regions of recirculation, flow separation, or uniform flow.

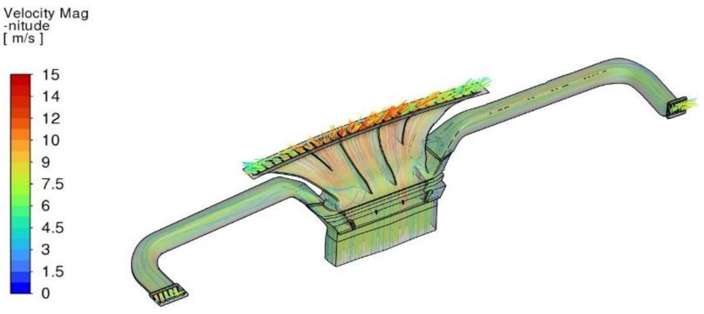

Fig no.2 Base case

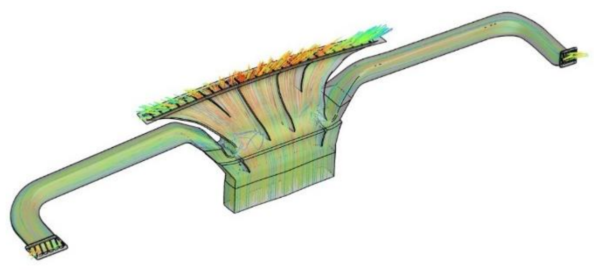

Fig no.3 Optimize case

4.2 Static pressure plot on duct surface of base case and optimize case.

A static pressure contour plot was generated on the duct surface to visualize pressure distribution, allowing identification of high-pressure zones and areas with significant pressure drop, which are critical for assessing flow efficiency and potential design improvements.

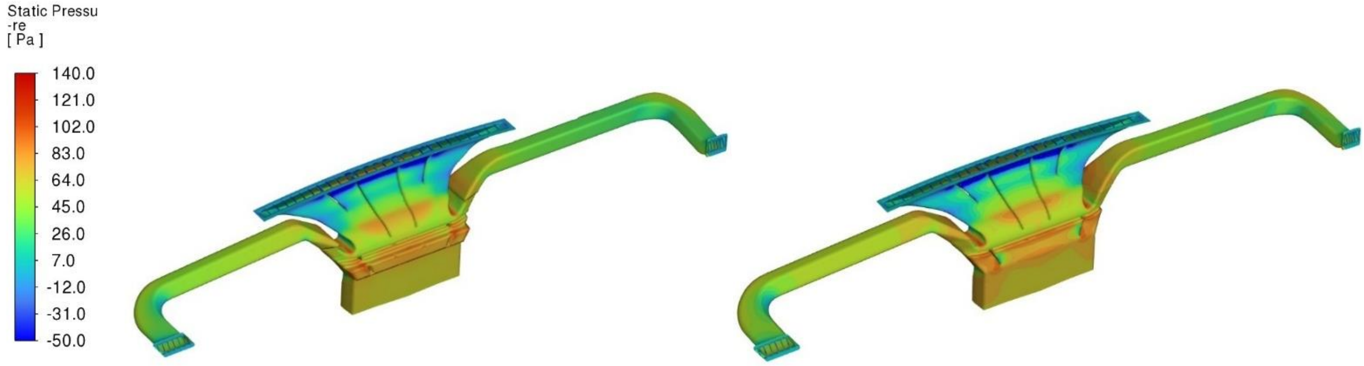

Fig no.4 Base case Fig no.5 Optimize case

4.3 Pressure drop result of base case and optimize case.

Sr.no	Duct parts name	Baseline case (pascal)	Optimize case (pascal)	Pressure difference (pascal)
1	Inlet pressure	100	100	0
2	Outlet pressure of central duct	47	46	1
3	Outlet pressure of left side duct	41	39	2
4	Outlet pressure of right side duct	38	35	3

4.4 Velocity contour on duct surface of base case and optimize case.

A velocity contour plot was created on the duct surface to illustrate the variation in airflow velocity throughout the system, helping to identify regions of high-speed flow, stagnation zones, and areas with non-uniform velocity distribution that could impact overall HVAC performance.

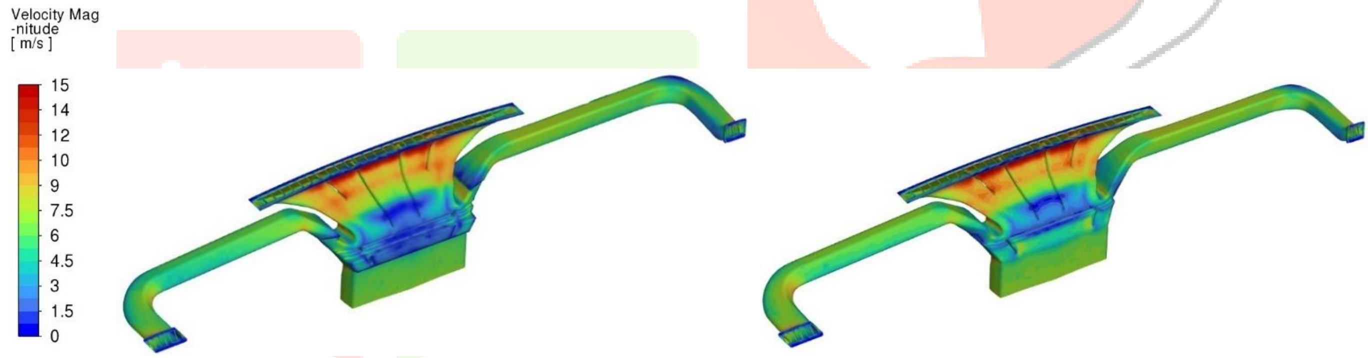

Fig no.4 Base case Fig no.5 Optimize case

5. CONCLUSION AND FUTURE WORK

5.1 Conclusion

Through the use of **Computational Fluid Dynamics (CFD) analysis**, the defrost duct geometry was systematically evaluated and optimized. By **modifying the duct geometry** to eliminate unnecessary features and improve flow paths, a **reduction of 6 Pascal in pressure drop** was achieved. This improvement confirms the effectiveness of CFD-driven design in enhancing airflow performance while maintaining the required packaging constraints. The study demonstrates that even small geometric refinements can lead to measurable efficiency gains in automotive HVAC systems.

5.2 Future Work Scope

While the current study successfully optimized the defrost duct and achieved a 6 Pascal reduction in pressure drop using CFD analysis, several opportunities remain for further improvement and expansion

5.2.1. Full HVAC System Optimization

The methodology used in this project can be extended to other duct branches, including floor and front vents, to improve the performance of the entire HVAC system, not just the defrost duct.

5.2.2. Mesh Optimization and Automation

Implementing automated meshing techniques and mesh optimization algorithms could further enhance simulation accuracy while reducing computational cost.

5.2.3. Validation Through Experimental Testing

To increase confidence in the CFD results, wind tunnel testing or flow bench experiments could be conducted to validate pressure drop and flow uniformity under real-world conditions

5.2.4. Material and Manufacturing Considerations

Future work may include the impact of material selection, wall roughness, and manufacturing tolerances on airflow performance, especially for ducts produced using advanced techniques like injection molding or 3D printing

ACKNOWLEDGMENTS

The author expresses sincere gratitude to Professor Dr. Anil Sahu for his invaluable guidance and consistent support throughout the project. Special thanks are extended to the Department of Mechanical Engineering at G. H. Raisoni College of Engineering & Management, Pune, for providing the necessary facilities and resources. The encouragement from faculty members, peers, and family members is also deeply appreciated.

REFERENCES

1.Florin Bode, Costin Coșoiu, Titus Joldos , Gabriel Mihai Sirbu, Paul Danca and Ilinca Nastase ,Impact of realistic boundary conditions on CFD simulations: A case study of vehicle ventilation *(Article Published in https://doi.org/10.1016/j.buildenv.2024.112264)*

2.Libin Tan and Yuejin Yuan Computational fluid dynamics simulation and performance optimization of an electrical vehicle Air-conditioning system *(Article Published in* https://doi.org/10.1016/j.aej.2021.05.001*)*

3.C.H. Chien, J.Y. Jang, Y.H. Chen and S.C. WU,3-d numerical and experimental analysis for airflow within a passenger compartment Published in International Journal of Automotive Technology, Vol. 9, No. 4, pp. 437−445 (2008)