IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)
An International Open Access, Peer-reviewed, Refereed Journal

Pressure Drop And Airflow Analysis Of An Automotive Air Conditioning Duct Through CFD Algorithm

Supreet Hulagur, 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 automotive air conditioning (AC) system plays a vital role in ensuring cabin comfort and vehicle endurance, and its design has received increasing attention. The air duct, which connects the AC compressor to the cockpit instrument panel outlets, significantly influences the system's cooling efficiency through its structural and flow characteristics. Its air delivery performance directly affects the uniformity of temperature and velocity fields within the cabin, thereby impacting occupant comfort. Due to spatial constraints, air duct design is among the most challenging aspects of vehicle HVAC system development. Consequently, optimizing the duct to balance spatial integration with energy efficiency is a key research focus. This study presents a CFD-based approach to the simulation and optimization of an automotive air duct. The model is initially developed using Ansa, meshed with Meshing software, and analyzed in Fluent to assess internal flow behavior. By applying defined inlet conditions, airflow distribution across outlets is evaluated. Based on a baseline vehicle duct model, issues such as turbulence and uneven outlet flow are addressed through design modifications—maintaining original spatial dimensions—and the improvements are validated through simulation results.

Index Terms – **AC duct, Automotive, CFD, Simulation, Model optimization and improvement.**

Introduction

For As of April 28, 2025, the global vehicle count has reached 1.475 billion, with annual production around 80 million units. This rapid growth poses significant challenges to global energy consumption, particularly due to the energy-intensive operation of automotive air conditioning systems. These systems, comprising components such as the compressor, condenser, evaporator, expansion valve, blower, and air ducts, rely on a closed-loop refrigerant cycle for temperature regulation. Among these, the air duct plays a critical role in determining pressure loss, airflow distribution, and thermal efficiency. Suboptimal duct design can cause vortex formation and uneven flow, reducing overall system performance and passenger comfort. This study proposes a CFD-based optimization approach for automotive air ducts. Using Ansa for modeling, meshing for discretization, and Fluent for flow simulation, the initial duct geometry is improved to enhance airflow and reduce energy loss. By adjusting the structure while maintaining design constraints, a more efficient duct configuration is achieved. To overcome the limitations of traditional parametric optimization constrained by CAD geometry, a mesh-based shape optimization algorithm is employed. This method, guided by the adjoint approach, enables localized geometry modifications to improve pressure and velocity fields at inlets and outlets efficiently. The result is a more energy-efficient and thermally effective air duct design, particularly suited for electric vehicle HVAC systems.

I. LITERATURE REVIEW

In automotive HVAC (Heating, Ventilation, and Air Conditioning) systems, the air conditioning ducts play a critical role in distributing conditioned air throughout the vehicle cabin. Two key parameters — pressure drop and airflow distribution — heavily influence system efficiency, passenger comfort, energy consumption, and noise levels. High pressure drop increases the blower load, leading to higher energy demands and potentially reduced airflow performance. Therefore, optimizing airflow and minimizing pressure drop are essential objectives in automotive duct design [1].

Pressure drop in ducts arises due to two main factors: frictional losses along the duct walls and dynamic losses caused by changes in geometry such as bends, expansions, contractions, and branches [2]. The Darcy-Weisbach equation is commonly used to describe frictional losses, while empirical loss coefficients are applied to account for geometric effects. Airflow in automotive ducts is typically turbulent, and adverse geometrical features can cause flow separation and secondary flows, contributing to increased pressure losses [3]. The use of Computational Fluid Dynamics (CFD) has become widespread for analyzing complex airflows and pressure behaviors in automotive ducts. CFD enables detailed visualization of velocity profiles, turbulence intensity, and pressure fields without the need for excessive physical prototyping.

Studies by Kim and Lee [4] demonstrated that the k-ω SST turbulence model provides better predictions of flow separation and pressure loss in ducts with sharp bends compared to the standard k-ε model. Ghosh et al. [5] emphasized the importance of fine meshing, particularly at bends and transitions, for accurate CFD results. Park et al. [6] validated CFD results against experimental measurements, showing good agreement and underscoring CFD's reliability in duct design optimization.

Experimental validation is crucial for confirming CFD predictions. Xu et al. [7] employed pressure taps and hot-wire anemometry to measure static pressure and velocity fields inside duct systems. Smoke visualization and Particle Image Velocimetry (PIV) methods are also used to study flow separation and turbulence characteristics. Experimental findings are often used to refine CFD models, ensuring that simulation results accurately reflect real-world conditions.

II. METHODOLOGY

3.1 CFD Modeling Principles for Automotive Air Conditioning Ducts

In the computational fluid dynamics (CFD) analysis of automotive air conditioning ducts, the primary focus is on the behavior of the cold airflow within the duct. To simplify the simulation without compromising accuracy, certain fluid properties—such as air compressibility, viscosity, and thermal conductivity—are neglected. As a result, the airflow is modeled as an incompressible, inviscid, and adiabatic fluid. Under these assumptions, the governing equation for the simulation is the continuity equation.

3.2 Turbulence Model Selection

The choice of an appropriate turbulence model depends on several factors, including fluid compressibility, the nature of the problem, required accuracy, available computational resources, and time constraints. Given the incompressible and adiabatic nature of the flow and the need for a balance between computational efficiency and predictive accuracy, the standard k–ε turbulence model was selected for this study. This model offers robust performance for internal flows and is well-suited for simulating airflow in HVAC duct systems.

III. PERFORMANCE ANALYSIS

4.1 Model preparation

In this study model preparation has been done in such way that getting already designed AC duct from internet (grabcad, shown in Figure 1) using it as base model for the pressure and velocity analysis. Once the simulations have been done then results were observed and based on the results observed optimization step has been done by making slight design improvement of the duct for better performance and air quality in the cabin compartment.

4.2 Mesh generation

This simulation experiment is based on the airflow analysis and pressure drop analysis of the air conditioning duct of a vehicle model. In this project Ansa is used as a Meshing software to carry out surface meshing operation in this model. It produced 70741 surface mesh elements (shown in Figure 2)

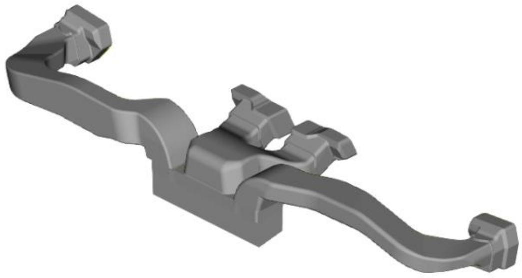


Figure 1. Duct geometry before meshing

Figure 2. Duct geometry after meshing

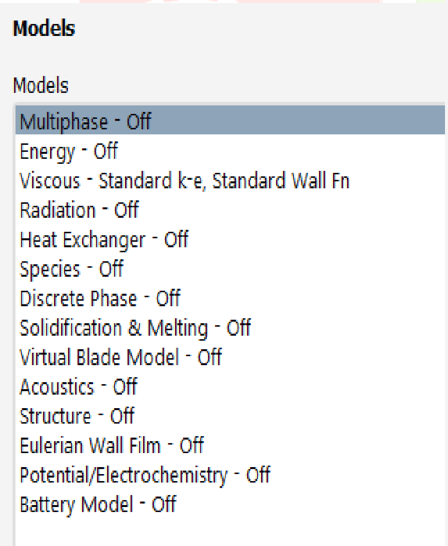
4.3 Setting of boundary conditions

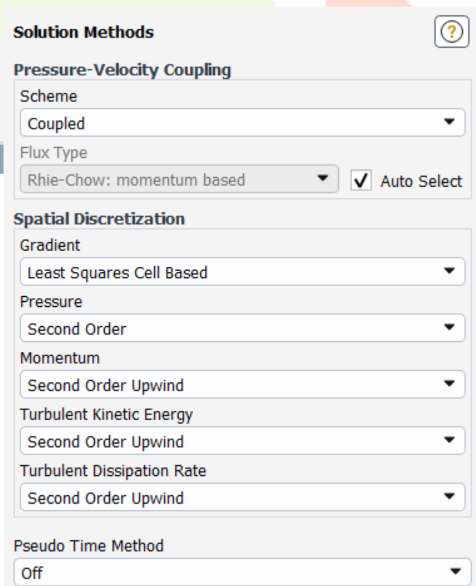
To improve computational efficiency and reduce modeling complexity in the simulation of automotive air-conditioning systems, structural simplifications are essential. Specifically, the evaporator and heater cores—characterized by complex louver fin geometries—are modeled as porous media. This approximation enables accurate prediction of pressure loss and heat transfer while significantly reducing mesh density and simulation time. For boundary condition setup, the system is defined with four outlet regions—designated as *Outlet-Left*, *Outlet-Middle Left*, *Outlet-Middle Right*, and *Outlet-Right*—representing the vehicle's front-facing air vents. The inlet is defined at the section connecting the air conditioning unit to the air duct. This configuration allows for efficient evaluation of airflow distribution and system performance within the simplified domain

- mass flow inlet is adopted, and its mass flow rate is 0.174333 kg / s.
- Outlet (all of them): pressure outlet is adopted, and its pressure value is set to 0 PA.
- Air conditioning box wall and air duct wall: adopt non sliding wall.

4.4 Setting of models

In fluid mechanics, different fluids have different characteristics, while the air fluid in the air duct of automotive air conditioning belongs to a relatively common fluid. According to its fluid properties and the characteristics of CFD software itself, the selection of its model is shown in Figure 4.3





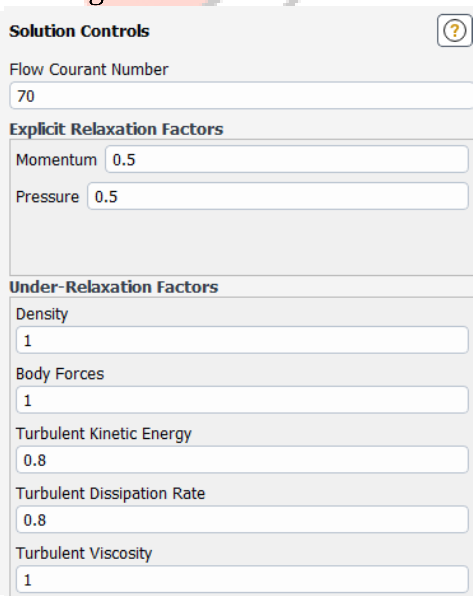


Figure 3. Solvers model selection **Figure 4.** Solvers solution methods **Figure 5.** Solvers solution controls

Where as inlet surafce is of inlet mass flow boundary type with a mass flow rate 1.74333 kg/s. Outlet surface are of pressure outlet boundary type with 0 PA pressure assigned to them. In this simulations coupled solution methods has been used (shown in figure 4) with default solution controls (shown in figure 5) which are widely used and adopted to solve the HVAC related solutions. The fluid full simulation operation was carried out with FLUENT software, the calculation steps are divided into 1000 steps in total, and the fluid velocity value has captured at every iteration at the outlet by defining the report definitions.

V. Results and Discussion

5.1. Display of simulation results and problems of the first version of the model

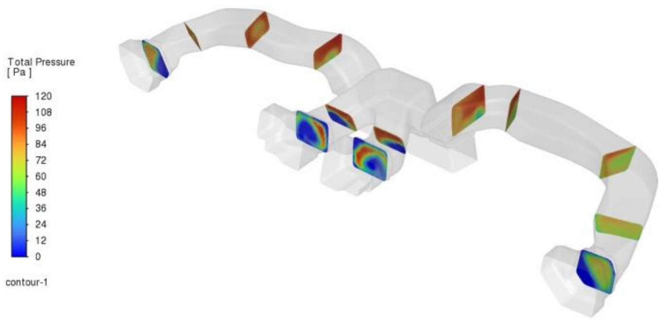
The fluid full simulation operation was carried out with FLUENT software, the calculation steps are divided into 1000 steps in total, and the fluid velocity streamline diagram of the whole pipeline was obtained, as well pressure drop contour plots on the specified cut planes are observed along the duct pipes.

The pressure drop on the different cut planes at different sections along the duct pipes is recorded and shown in Table 1.

Left duct pipe	Cut plane left 1	95.6914
	Cut plane left 2	90.5128
	Cut plane left 3	84.57366
	Cut plane left 4	40.5945
Middle left duct pipe	Cut plane mid left 1	46.2172
	Cut plane mid left 2	28.8159
Middle right duct pipe	Cut plane mid right 1	58.5552
	Cut plane mid right 2	40.9951
Right duct pipe	Cut plane right 1	87.2793
	Cut plane right 2	81.189
	Cut plane right 3	72.8159
	Cut plane right 4	68.4478
	Cut plane right 5	36.8690

Table 1. Calculation results of pressure drop along the original air duct (Pa)

Pressure and velocity distribution on the specified cut planes are shown in Figure 6 & 7.



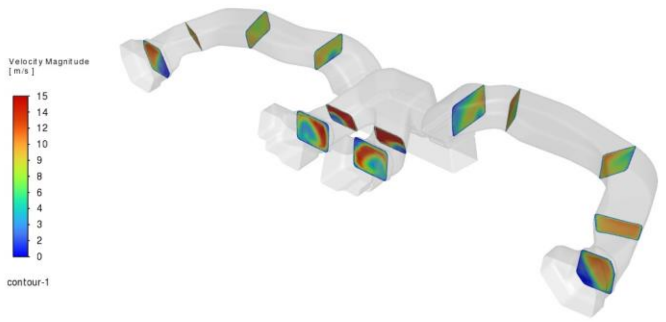


Figure 6. Pressure distribution plot on original duct **Figure 7.** velocity distribution plot on original duct

The average wind speed of the four air outlet pipes is recorded and shown in Table 2.

Original air duct	Average airflow at outlet
Left	4.1287
Middle-left	4.9918
Middle-right	5.5604
Right	4.1882

Table 2. Calculation results of average wind speed at the outlet of the original air duct (m/s)

Velocity distribution inside the duct pipe from inlet to all 4 outlets are shown (in below Figure 8) using streamlines.

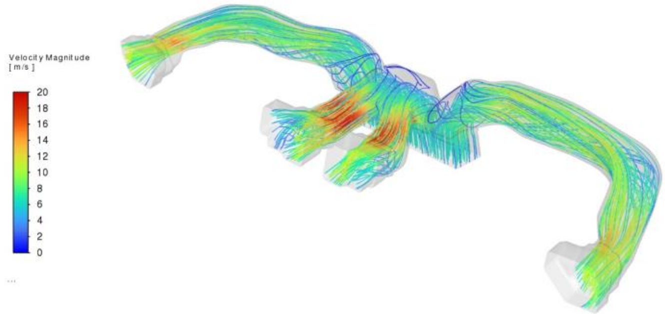


Figure 8. velocity distribution plot on original duct

5.2. Optimization and adjustment of air duct structure

Generally, it is not allowed to affect the interior decoration layout in the optimization of the air conditioning duct structure, so this optimization retains the basic structure of the original air duct, including the location and size of the air outlet and inlet. By observing the velocity streamline diagram of the original air duct (shown in figure 11), it is found that the below highlighted area (shown in figure 9) is bulged kind of structure where it is observed that low velocity region shown in figure 12 in which duct has been cut at bulged area and can see low velocity in cut plane contour after base duct simulation. The low velocity region inside the duct further creates detached air flow from the inside duct surface. so, the structure of the right branch air ducts is optimized (shown in Figure 10)

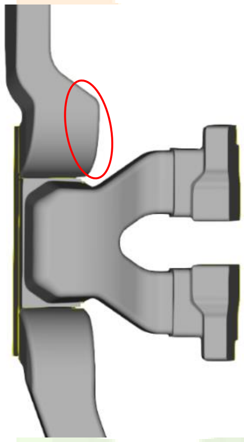


Figure 9. Before the change

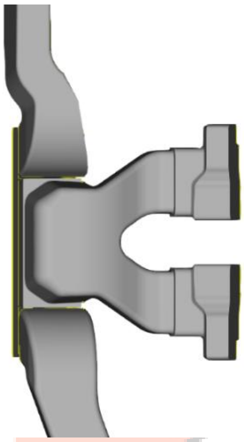


Figure 10. After the change

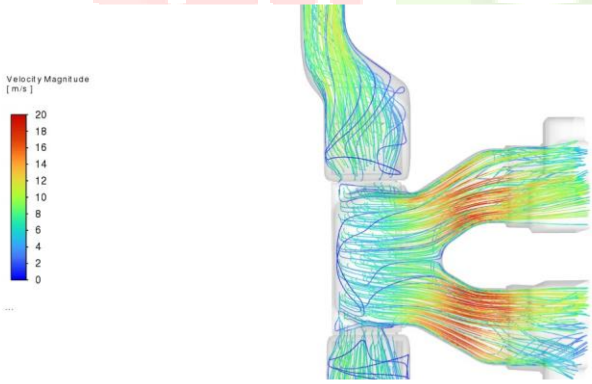


Figure 11. velocity streamline distribution top view

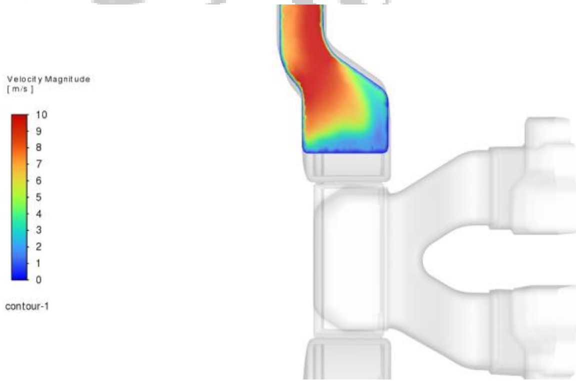


Figure 12. velocity distribution on cut plane on right duct

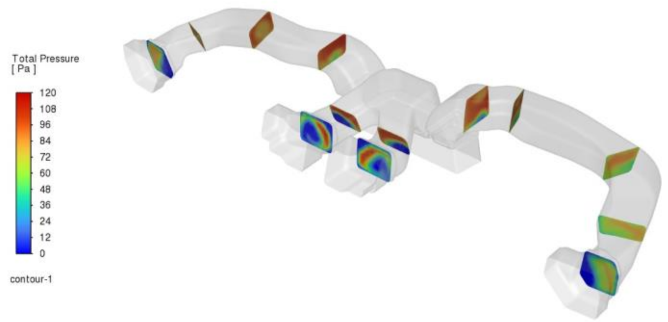
5.2.1. Simulation results of the optimized model

First, the software platform mentioned above was used to mesh the new model. Then the meshing model was put into the Fluent software and carried out volume meshing and simulation operations, and the selection of boundary conditions and models was the same as before. It can be seen from the improved air duct pressure distribution on the cut plots is more uniformed. The pressure values on cut planes along the duct pipes are recorded and tabulated in below Table 3.

Left duct pipe	Cut plane left 1	92.9366
	Cut plane left 2	87.42624
	Cut plane left 3	81.2797
	Cut plane left 4	40.2183
Middle left duct pipe	Cut plane mid left 1	50.3048
	Cut plane mid left 2	34.7584
Middle right duct pipe	Cut plane mid right 1	56.7007
	Cut plane mid right 2	41.7750
Right duct pipe	Cut plane right 1	86.0653
	Cut plane right 2	78.8547
	Cut plane right 3	71.4945
	Cut plane right 4	67.385
	Cut plane right 5	39.0441

Table 3. Calculation results of pressure drop along the optimized air duct (Pa)

Pressure and velocity distribution on the specified cut planes are shown in Figure 13 & 14.



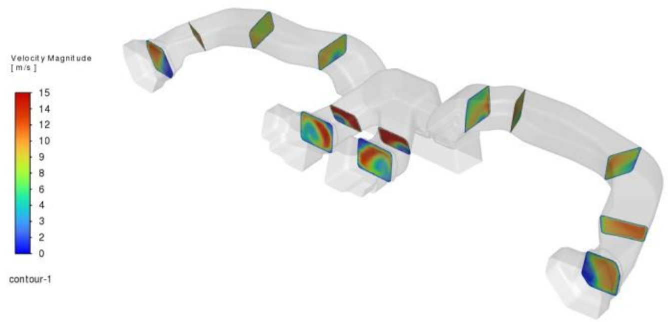


Figure 13. Pressure distribution plot on optimized duct **Figure 14.** velocity distribution plot on optimized duct

The average wind speed of the four air outlet pipes is recorded and shown in Table 4.

Optimized air duct	Average airflow at outlet
Left	4.1280
Middle-left	5.2622
Middle-right	5.4712
Right	4.2396

Table 4. Calculation results of average wind speed at the outlet of the optimized air duct (m/s)

It can be clearly seen from the improved air duct streamline diagram that compared with the first version of the air duct before, the uniformity of the internal streamline of the right duct pipe has been greatly improved. The velocity distribution cloud diagram of the four outlet pipelines is shown in Figure 15. The new average wind speed of the four air outlet pipes is recorded and shown in Table 4. By comparing the velocity distribution plots and average velocity table of the four air outlets of the original model and the new model, it can be clearly seen that the fluid velocity of right duct pipe of the improved model is more average. Therefore, it can be considered that the model improvement scheme is successful.

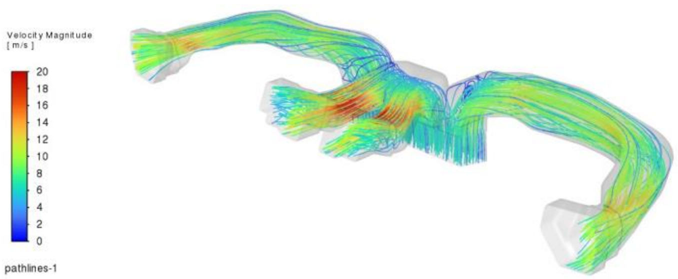


Figure 15. velocity distribution plot on optimized duct

VI. Conclusion and Future scope

This study investigates the airflow characteristics and pressure drop in the front air-conditioning duct of a vehicle using Computational Fluid Dynamics (CFD). The duct geometry was initially modeled, meshed using Meshing software, and analyzed in FLUENT. The simulation results revealed two key issues: uneven airflow distribution in the right-side duct and turbulence-induced flow disturbances at the joint region. To address these issues, a structural optimization of the right duct was performed using CFD techniques. The excessive bulged surface of the right duct was trimmed, ensuring that the overall dimensions remained compatible with real-world installation constraints. Comparative analysis of the original and optimized models—including streamline visualizations, pressure drop profiles, and velocity distribution contours at the four outlets (Outlet-Left, Outlet-Middle Left, Outlet-Middle Right, Outlet-Right)—demonstrates that the modified design effectively mitigates the previously identified flow disturbances. Moreover, under identical inlet conditions, the optimized duct not only achieves a lower pressure drop but also enhances the average outlet air velocity across all four outlets, thereby improving overall system performance and passenger comfort.

The future scope in this domain includes several promising directions such as Future CFD studies will increasingly incorporate human thermal comfort models such as PMV (Predicted Mean Vote) and PPD (Predicted Percentage of Dissatisfied) to evaluate duct performance from the perspective of passenger comfort, enabling more precise climate control systems. Future CFD optimization will extend beyond the duct itself to encompass the entire HVAC and cabin environment. This includes considering the effect of external climate conditions, cabin geometry, and real-time user inputs on duct performance.

References

[1] Munson, B. R., Young, D. F., & Okiishi, T. H. Fundamentals of Fluid Mechanics, 7th ed., Wiley, 2013.

[2] Park, J., Lee, H., & Kim, Y. "Optimization of Air Duct Systems Using CFD in Automotive Applications," SAE International Journal of Passenger Cars-Mechanical Systems, vol. 7, no. 3, pp. 897–904, 2014.

[3] Ghosh, S., Shukla, P. C., & Ghosh, A. "CFD Based Analysis and Optimization of Automotive HVAC System for Air Flow Improvement," Applied Thermal Engineering, vol. 147, pp. 1041–1053, 2019.

[4] Kim, Y., & Lee, K. "Comparison of Turbulence Models in CFD Simulation of Automotive HVAC Systems," *SAE Technical Paper*, 2016-01-0207, 2016.

[5] Ghosh, S., Shukla, P. C., & Ghosh, A., *ibid.*

[6] Park, J., Lee, H., & Kim, Y., *ibid.*

[7] Xu, M., Tan, H., & Lu, Y. "Experimental and Numerical Study of Flow Characteristics in Automotive Air-Conditioning Ducts," *Energy and Buildings*, vol. 92, pp. 50–59, 2015.