International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 11 Issue: 12 | Dec 2024 www.irjet.net p-ISSN: 2395-0072

Aerodynamic Analysis of Flow Over a 2D Sports Car using Computational Fluid Dynamics

Prof. Sathish H[1], Anish Rajendra Gaikwad[2], Darshan T V[3], Gokul R[4], Mohammed Rayan Habeeb[5]

[1]Assistant Professor, Dept. of Mechanical Engineering, MIT Mysore, Karnataka, India
[2-5]Undergraduate students, Dept. of Mechanical Engineering, MIT Mysore, Karnataka, India

---

Abstract - *This study explores the aerodynamic performance of a 2D sports car model using Computational Fluid Dynamics (CFD). A wind tunnel simulation with a 12 m × 45 m test section modeled airflow around the car at 20 m/s. The investigation included geometry definition, structured meshing, and numerical analysis in ANSYS Fluent to assess static pressure and velocity contours, drag forces, and areas for design optimization. Results indicated high-pressure zones at the front stagnation point and low-pressure regions over the roof, consistent with streamlined airflow. Velocity contours revealed accelerated flow over the roof and wake regions at the rear, highlighting flow separation and potential areas for improvement. The drag coefficient converged at 0.012, while the drag force stabilized at 70 N, showcasing the car's aerodynamic efficiency. The study demonstrates the effectiveness of CFD in optimizing vehicle design by reducing drag and improving performance. These findings provide valuable insights into fuel efficiency, speed, and stability enhancements for high-performance sports cars, laying a foundation for future innovations in automotive aerodynamics.*

Key Words: External flows, CFD, drag force, drag coefficient, 2D-Sportscar, Aerodynamics, Computational Fluid Dynamics.

1.INTRODUCTION

In the modern automotive industry, achieving superior aerodynamic performance is paramount for optimizing vehicle efficiency, speed, and sustainability. This paper delves into the computational study of airflow around a two-dimensional (2D) sports car model, employing Computational Fluid Dynamics (CFD) techniques. CFD serves as a pivotal tool, enabling the analysis and simulation of fluid flow phenomena that are often challenging to replicate through physical experiments.

The study's primary focus is on analyzing aerodynamic characteristics, including drag coefficient, drag force, and flow patterns around the car model. Utilizing tools such as ANSYS Fluent, a comprehensive simulation was conducted in a virtual wind tunnel environment. This setup included a streamlined car geometry, well-defined boundary conditions, and turbulence modeling to ensure accurate predictions of real-world behavior.

Key outcomes of the research demonstrate the effectiveness of CFD in identifying and mitigating aerodynamic inefficiencies. By visualizing flow patterns and quantifying aerodynamic forces, this work provides actionable insights for enhancing the design of high-performance vehicles. The findings contribute to the broader goal of reducing aerodynamic resistance, thereby advancing the fields of energy efficiency and automotive engineering.

2. COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) has emerged as a transformative tool in modern engineering and science, enabling the simulation and analysis of complex fluid flow phenomena with remarkable precision. By solving the governing equations of fluid motion—such as the continuity, momentum, and energy equations—CFD provides a computational framework to model the behavior of fluids in diverse scenarios, from aerospace and automotive applications to biomedical and environmental engineering.

The core strength of CFD lies in its ability to address challenges that are either impractical or cost-prohibitive to explore experimentally. Using numerical methods and computational algorithms, it transforms physical domains into discrete grids or meshes, where fluid properties such as pressure, velocity, and temperature are computed. This process offers insights into intricate flow patterns, turbulence characteristics, and heat transfer mechanisms, guiding the optimization of designs and processes.

Despite its advantages, CFD is not without challenges. Accurate simulations demand high-quality mesh generation, careful selection of turbulence models, and the implementation of precise boundary conditions. Furthermore, the computational cost associated with high-fidelity simulations remains a significant consideration, particularly for problems involving large-scale or highly dynamic flows.

This paper explores the application of CFD in analyzing aerodynamic properties of a two-dimensional sports car model, emphasizing its role in predicting drag forces, visualizing flow behavior, and optimizing vehicle performance. The study highlights the versatility of CFD in bridging the gap between theoretical fluid mechanics and

practical engineering applications, underscoring its indispensability in contemporary research and development.

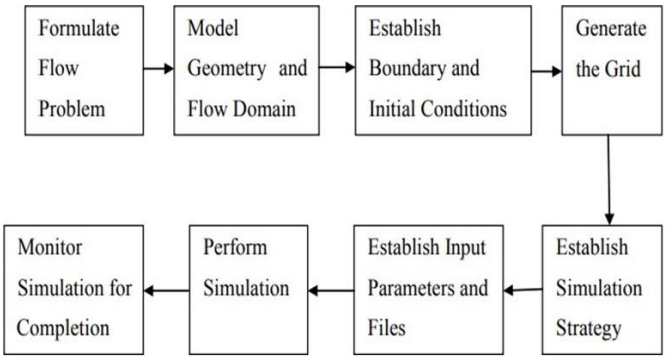


Fig -2.1: Steps involved in CFD

2.1 About the project

The project involves simulating airflow over a 2D cross-section of a sports car traveling at 72 km/h. Using specialized software (Ansys Fluent), the simulation setup includes importing pre-defined mesh files and performing an analysis of drag forces of sports car. This approach highlights the potential for design optimization, enabling the development of vehicles with reduced aerodynamic resistance and enhanced performance

3. PROBLEM DEFINITION

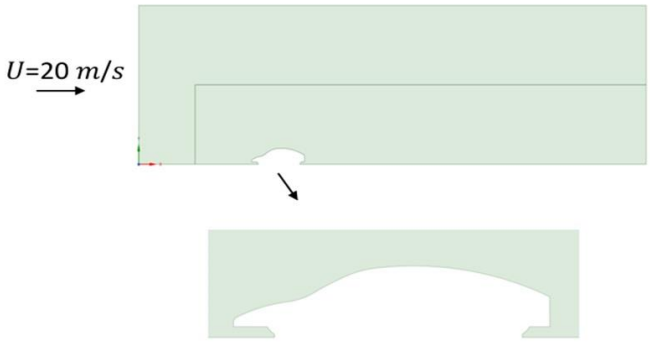


Fig -2.1: Problem description

The wind tunnel model setup comprises a 12m x 45m test section designed to simulate real-world wind conditions. A reference car model with dimensions of 5 meter in length, 2.34 m² frontal area, and 1.8 meters depth is placed within this section. This configuration enables the study of aerodynamic characteristics like drag and lift. The wind tunnel data, coupled with CFD simulations, provides valuable insights for automotive engineers to refine vehicle designs and improve overall aerodynamic performance.

3.1 Assumptions

- **Incompressible flow:** Since for inlet velocity of 20 m/s which corresponds to Mach number of 0.06, effects of air compressibility are, in general, negligible when Mach number is less thank 0.3
- **Turbulent flow:** Since the Reynolds number is Re = 6.4*10^6
- **Steady flow:** The variables like velocity, pressure, density, and temperature remain constant at a fixed location as time progresses

3.2 Fluid Properties

- **Density of air**, ρ_air = 1.225 kg/m^3
- **Viscosity of air**, μ_air = 1.7894 ×10^-5 kg/m-s
- Mach Number is 0.06 corresponds to inlet velocity of 20 m/s

3.3 Boundary conditions

The various boundary conditions are as follows:

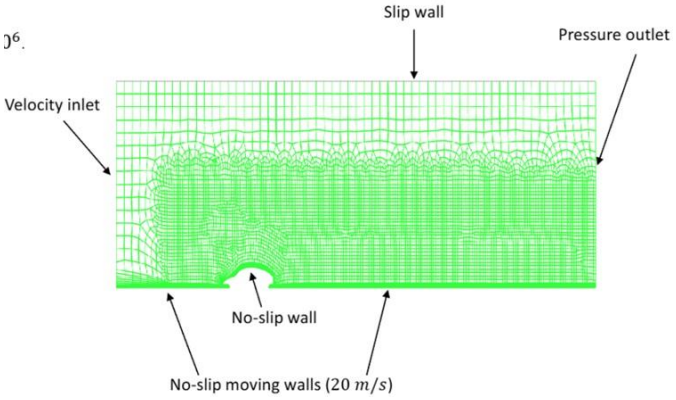


Fig -3.3: Boundary conditions

Table -3.3: Boundary patches and conditions

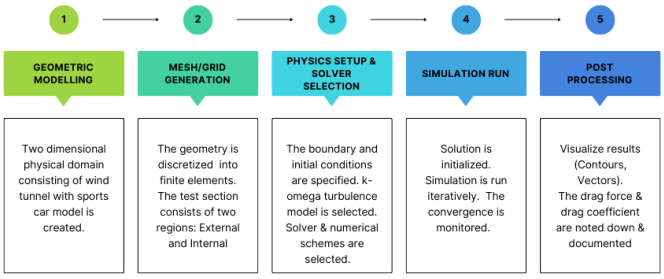
Sl. No.	Boundary	Condition
1	Inlet	*velocity inlet*
2	Outlet	*pressure outlet*
3	Wall-tunnel	*slip wall*
4	Car	*no-slip wall*
5	Grounds	*moving wall*
6	Interiors	*interior*

4. OBJECTIVES

The primary objective of this mini-project is to conduct a comprehensive numerical simulation of airflow around a car model in a wind tunnel. This analysis aims to:

1. **Validate the Numerical Model:** Ensure the accuracy of the simulation by checking mass conservation (continuity equation).
2. **Visualize Flow Patterns:** Generate contour plots of static pressure and velocity magnitude to understand the flow field around the car.
3. **Analyze Flow Vectors:** Plot velocity vectors to visualize the direction and magnitude of flow at different points.
4. **Quantify Aerodynamic Drag:** Calculate pressure drag, viscous drag, coefficient of drag, and the total drag force acting on the car

5. METHODOLOGY



6. CFD ANALYSIS

6.1 Geometric Modelling

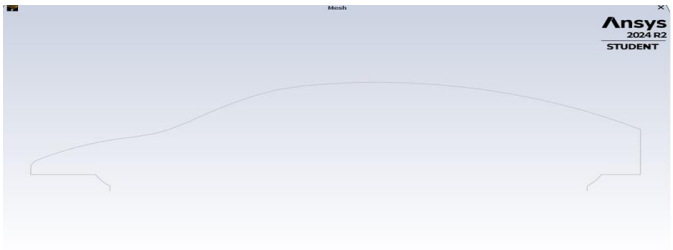


Fig -6.1: 2D Sports car geometry

The wind tunnel setup features a test section measuring 12 meters in width and 45 meters in length, specifically designed to replicate real-world wind conditions. Within this section, a reference car model is positioned, having dimensions of 5 meters in length, a frontal area of 2.34 m^2, and a depth of 1.8 meters. After the creation of Geometric model using the required coordinates, creation of parts is done corresponding to the boundaries. Association of the entities to the geometry is also Done.

6.2 Mesh/Grid Generation

The whole of the test section consists of two regions:

External:

Where mesh is less refined (Consisting of coarse elements)

Internal:

The mesh is more refined (Consisting of fine elements)

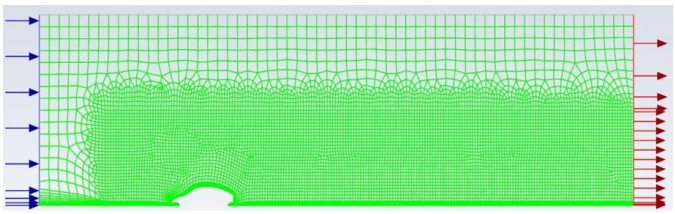


Fig -6.2.1: Mesh generation

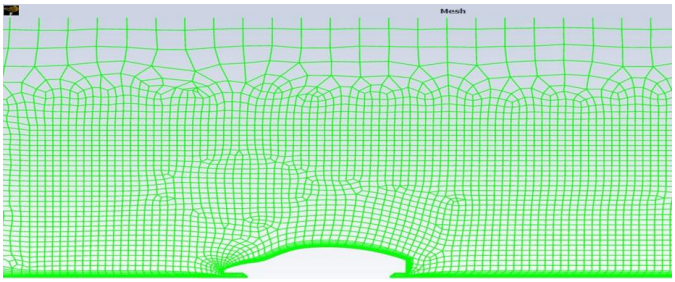


Fig -6.2.2: Mesh generation ~ zoomed view

7. Results and Discussions:

The results consist of residuals which quantify how far the current solution deviates from satisfying the governing equations. The Velocity magnitude contours visualize the distribution of flow speed around a 2D car, highlighting acceleration zones, separation regions, and boundary layer effects critical for understanding aerodynamic performance. The Pressure contours of 2D car reveal areas of high and low static pressure, indicating stagnation points, pressure gradients, and recovery zones that directly impact lift, drag, and flow stability. The velocity vectors illustrate the direction and magnitude of flow, providing insight into flow patterns, recirculation zones, and boundary layer behavior around the 2D car model. Also results involves the calculation of drag force, quantifies the aerodynamic resistance experienced 2D sports car, while the drag coefficient is a dimensionless measure of its efficiency in minimizing drag relative to its shape and flow conditions.

7.1 Residuals

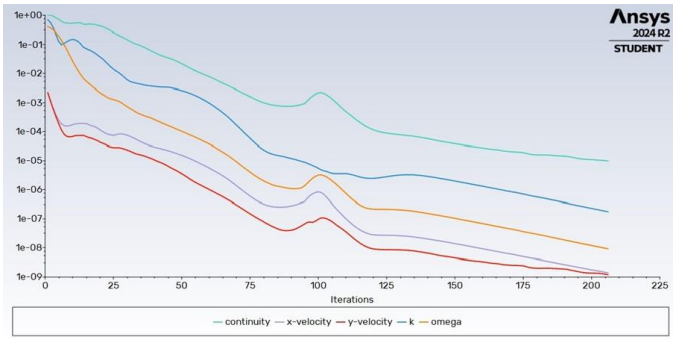


Fig -7.1: Residuals

The residuals of continuity, x-velocity, y-velocity, k and omega are as shown in the Fig-7.1: The observations from the residual plots are as follows.

- *General Trend:* All residuals (continuity, x-velocity, y-velocity, k and omega) decrease over iterations, which indicates the solution is converging.
- *Continuity Residuals:* The continuity residual (light blue line) decreases but stabilizes at a higher value than the other parameters. This may indicate slight mass conservation issues or that further refinement is needed for better convergence.
- *Velocity Components:* Residuals for x-velocity (orange line) and y-velocity (red line) drop significantly and stabilize below, this suggests that momentum equations are well resolved.
- *Turbulence Parameters (k and omega):* The turbulence parameters (k and omega) also converge well. Stabilizing below. This indicates good resolution of the turbulence model.
- *Number of Iterations:* Converges: Convergence appears to occur around 175 to 200 iterations, indicating reasonable computational setup.

7.2 Velocity Magnitude Contours

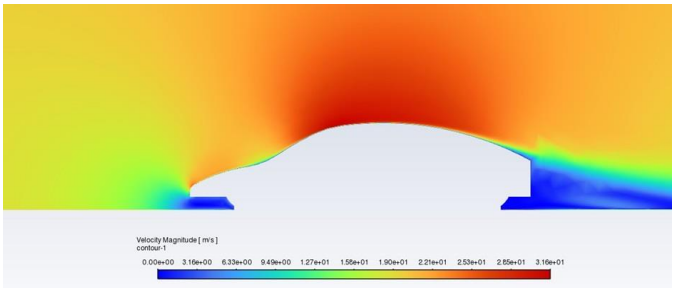


Fig -7.2: Velocity magnitude contours

The velocity magnitude contours around the 2D sports car are shown in Fig-7.2. The observations from the velocity magnitude contours are as follow.

- *High velocity region over the roof:* The red to orange color above the car indicates airflow acceleration over the roof. This suggests a streamlined flow but might also cause lift, depending on the pressure distribution.
- *Flow separation at the rear:* The blue/green region at the rear of the car could indicate flow deceleration and potential separation, which increases drag. This separation zone suggests opportunities to optimize the car's rear design (example adding a diffuser or optimizing the tail shape)
- *Low velocity region near the ground:* The blue region near the base of the car shows how velocities due to boundary layer effects. This might lead to increased pressure drag if not carefully designed.
- *Gradient and transition zones:* Observe the smooth color transition from green to orange or red around the front hood and windshield. This indicates gradual velocity changes, which is favorable for reducing the drag.

7.3 Velocity Vectors

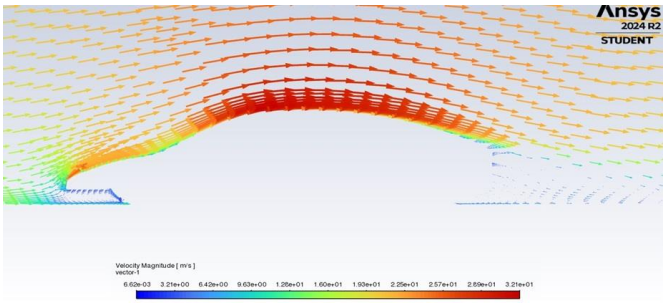


Fig -7.3: Velocity vectors

The velocity vectors around the 2D sports car are shown in Fig-7.3. The observations from the velocity vectors are as follow.

- *Accelerated Flow over the Roof (Red Region):* The airflow accelerates significantly over the roof of the car, as indicated by the densely packed and elongated vectors with high velocity (red region). This is a characteristic of streamlined flow over a curved surface, reducing pressure (as per Bernoulli's principle).
- *Stagnation at the Front (Blue Region):* The blue region at the front of the car shows minimal velocity due to stagnation. This corresponds to the high-pressure zone observed in the static pressure plot.
- *Wake Region at the Rear:* Behind the car, vectors become less uniform and exhibit recirculation zones, indicating

flow separation and wake formation. This wake contributes to drag and is an area for aerodynamic optimization.

- *Ground Flow Interaction:* Near the ground and the car's base, airflow vectors indicate boundary layer effects. Velocity decreases near the surface due to friction and the no-slip condition.
- *Flow Direction Consistency:* Overall, the vector field shows well-directed flow with no significant turbulence except in the rear wake, indicating a relatively clean aerodynamic profile.

7.4 Static Pressure Contours

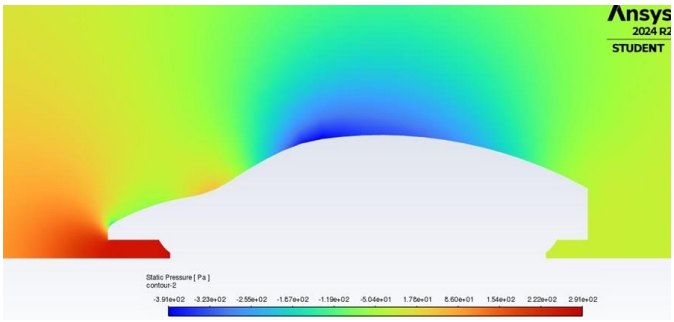


Fig -7.4: Static pressure contours

The static pressure distribution around the car exhibited expected trends. Observations from the static pressure plot are as follows:

- *High pressure region was observed at the front stagnation point of the car (red zone):* The red region at the car's front indicates a stagnation zone where the airflows slow down and pressure increases. This contributes significantly to the drag force.
- *Low pressure region over the roof (blue zone):* The blue zone indicates air flow acceleration over the car, reducing static pressure.
- *Pressure gradient transition from the front to rear:* The gradual transition from high to low pressure along the cars body suggests smooth airflow attachment.
- *Rear pressure zone:* The green/yellow region at the rear indicates moderate pressure recovery.
- *Ground pressure:* The consistent pressure near the base of the car indicates the boundary layer effect.

7.5 Drag Coefficient

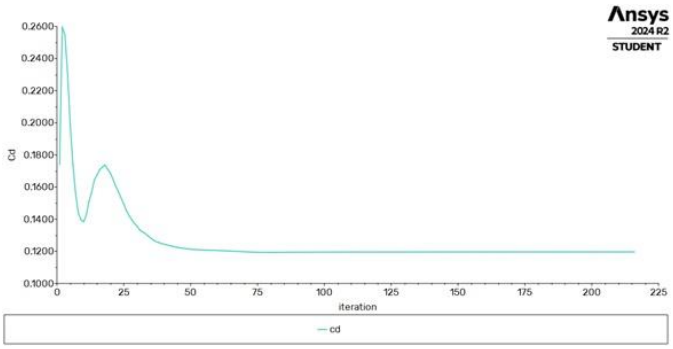


Chart -7.5: Drag coefficient

- *Initial Oscillations:* The drag coefficient starts at a high value (~0.26) and shows oscillations during the early iterations. This behavior indicates the solver adjusting to initial flow conditions.
- *Convergence:* After around 50 iterations the oscillations reduce significantly, and stabilizes. By approximately 200 iterations, the simulation reaches a steady state.
- *Final Drag Coefficient:* The converged value is approximately 0.02, representing the car's aerodynamic drag under steady conditions.
- *Aerodynamic Performance:* The drag coefficient of 0.02 is very low, highlighting the streamlined design of the 2D sports car model. This indicates excellent aerodynamic efficiency, reducing drag forces.
- *Practical Insight:* Low drag improves vehicle speed, stability, and fuel efficiency, which are crucial for sports car performance.

7.6 Drag Force

The Drag force Vs. number of iterations is as shown in the Chart 7.6.

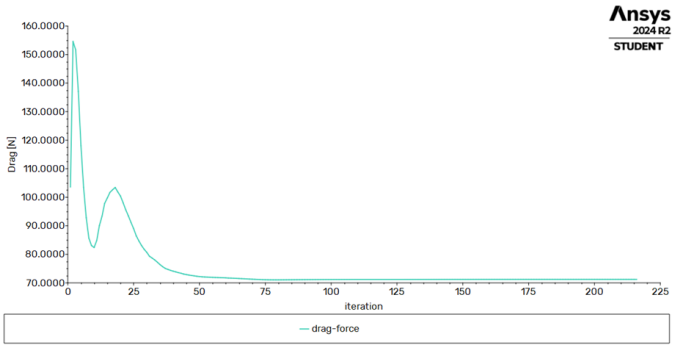


Chart -7.6: Drag force

- *Initial Oscillations:* The drag force starts at a high value (~160N) and shows significant fluctuations in the early iterations.
- *Convergence:* After approximately 50 iterations, the oscillations diminish, and the drag force stabilizes. By around 200 iterations, the solution converges to a steady value.
- *Final Drag Force:* The converged drag force is approximately 70N, representing the aerodynamic resistance experienced b the 2D sports car model.
- *Relation to Aerodynamics:* The reduction in drag force highlights the streamlined design's effectiveness in minimizing aerodynamic resistance.
- *Practical Insight:* A lower force directly improves vehicle performance by enhancing fuel efficiency, speed, and stability. This is particularly crucial for sports cars to achieve high speed efficiency.

REFERENCES

[1] Y. A. Cengel and J. M. Simbala, *"Fluid Mechanics – Fundamentals & Applications"* 2nd ed., McGraw-Hill, 2010.

[2] H Sathish and K J Yogesh Kumar, *"Computational Fluid Dynamics - Laboratory Manual,"* MIT Mysore, unpublished.

[3] R. K. Bansal, *"Fluid Mechanics & Hydraulic Machines"* 9th ed., Laxmi Publications, 2005.

[4] B. R. Munson and D. F. Young, *"Fundamentals of Fluid Mechanics"* Wiley, 2013.

[5] W.-H. Hucho, *"Aerodynamics of Road Vehicles, from Fluid Mechanics to Vehicle Engineering"* Butterworth & Co. Ltd., 1987.

[6] J. D. Anderson, *"Fundamentals of Aerodynamics"* McGraw-Hill, 2001.

[7] ANSYS, *"ANSYS Tutorials"* version 19.2, Canonsburg, PA, ANSYS Inc., 2018.

[8] M. Peric and J. H. Ferziger, *"Computational Methods for Fluid Dynamics"* Springer-Verlag, 1996.