

Design, Fabrication and Experimental Validation of an Open Channel Water Tunnel for Ahmed Body Aerodynamic Analysis using CFD and Experiments.

Mohd Mustufa Ansari, Shaikh Abdul Basit, Rauhaan Pawaskar, Safwan Faki, Prof. Shreyas Lokhande

1,2,3,4 UG Students, Department of Mechanical Engineering, INDIA

5, Professor of Department of Mechanical Engineering Rizvi College of Engineering Mumbai, INDIA

Abstract:- The present study focuses on the design, fabrication, and validation of an open channel water tunnel for analyzing flow behavior around an Ahmed body using both experimental and computational approaches. The Ahmed body, a standard benchmark in automotive aerodynamics, is used to investigate wake formation, flow separation, and drag characteristics. The experimental setup is designed to ensure uniform and steady flow conditions, enabling effective flow visualization through dye injection techniques. In parallel, Computational Fluid Dynamics (CFD) simulations are performed using the $k-\epsilon$ turbulence model to predict velocity distribution, pressure contours, and aerodynamic coefficients. A comparative analysis between experimental observations and numerical results demonstrates good qualitative agreement, validating the effectiveness of the developed setup. The study highlights the feasibility of a low-cost water tunnel as a reliable tool for aerodynamic analysis and provides insights into bluff body flow behavior relevant to automotive applications..

I. INTRODUCTION

Aerodynamic analysis of bluff bodies plays a crucial role in improving vehicle performance and fuel efficiency. The Ahmed body is widely used as a simplified model to study complex flow phenomena such as flow separation, wake formation, and drag characteristics. Experimental setups like wind tunnels are expensive, making water tunnels a cost-effective alternative. This study aims to design and develop an open channel water tunnel and validate its performance using CFD analysis. The objective is to create a low-cost yet efficient experimental setup capable of replicating real flow conditions and validating numerical simulations.

3. Techniques Developed

Flow Visualization Technique: Dye injection method is used to track streamlines and identify separation zones. **Uniform Flow Development:** Honeycomb structures and screens are implemented to reduce turbulence and ensure laminar inlet

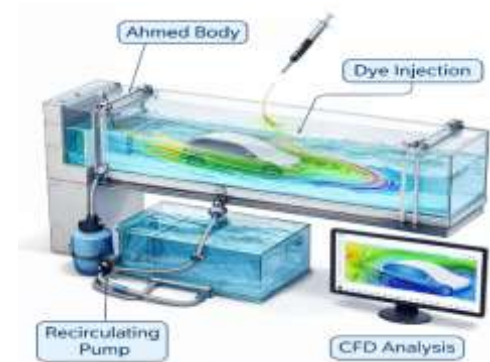


Fig.1:- AI Image for representation

II. Experimental Investigations

1. Experimental Setup

The experimental setup consists of an open channel water tunnel designed and fabricated to study flow behavior around an Ahmed body model. The setup includes a water reservoir, pump, flow straightener, test section, and return channel. The test section is made of transparent material (such as acrylic) to enable clear visualization of flow patterns. A scaled Ahmed body model is placed inside the test section. The flow is generated using a centrifugal pump, and uniform flow conditions are achieved using honeycomb structures and screens at the inlet. Flow visualization techniques such as dye injection are used to observe separation regions, vortex formation, and wake structures.

2. Experimental Procedure

The following procedure is adopted for conducting the experiments:

1. The water tunnel is filled and the pump is started to establish steady flow.
2. Flow rate is adjusted to achieve the desired Reynolds number.
3. The Ahmed body model is positioned properly in the test sections.

conditions. Low-Cost Design Approach: The water tunnel is designed using locally available materials to reduce overall cost while maintaining functionality.

4. Methodology Adopted

The methodology integrates both experimental and computational approaches:

a) Experimental Method

- Conduct flow visualization experiments using the open channel water tunnel.
- Record qualitative observations of flow behavior around the Ahmed body.
- Identify wake regions and vortex structures.

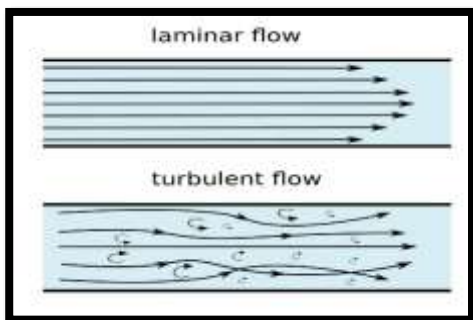
b) CFD Method

- Create a 3D model of the Ahmed body using CAD software.
- Generate computational mesh and define boundary conditions.
- Apply appropriate turbulence models (e.g., $k-\epsilon$ or $k-\omega$ SST).
- Perform simulations to obtain velocity, pressure, and drag characteristics.

c) Validation

- Compare experimental observations with CFD results.
- Analyze similarities and deviations in wake formation and flow separation.
-

III. TYPES OF FLOW



IV. METHODOLOGY & WORKING PRINCIPLE

The present study adopts a combined experimental and computational approach to investigate the aerodynamic behavior of flow over an Ahmed body using an open channel water tunnel. The methodology is structured to ensure accurate flow visualization, controlled experimental conditions, and reliable validation using Computational Fluid Dynamics (CFD).

The experimental setup consists of a specially designed open channel water tunnel fabricated using transparent acrylic material to facilitate clear visualization of flow patterns. The system includes a water reservoir, centrifugal pump, flow straightening section, test section, and return channel. Uniform flow at the inlet is achieved by incorporating honeycomb structures and fine mesh screens, which help

reduce turbulence intensity and align the flow direction before entering the test section.

A scaled Ahmed body model is mounted securely at the center of the test section to minimize wall interference effects. The flow within the tunnel is generated using a pump, and the velocity is controlled to achieve the desired Reynolds number. Flow visualization is carried out using dye injection techniques, where a colored dye is introduced upstream of the model to trace streamlines and identify flow separation, recirculation zones, and wake formation behind the body. Multiple experimental trials are conducted at different flow velocities to ensure repeatability and consistency of results.

In parallel, a computational analysis is performed using ANSYS Fluent to simulate the flow behavior under similar boundary conditions. A two-dimensional model of the Ahmed body is created, and an appropriate computational mesh is generated with refined elements near the body surface and wake region to capture flow gradients accurately. The simulations are carried out using the standard $k-\epsilon$ turbulence model, which is widely used for external flow analysis. Boundary conditions are defined as velocity inlet and pressure outlet, with no-slip conditions applied at solid walls.

The governing equations of fluid flow, including the continuity and Navier–Stokes equations, are solved using a pressure-based solver. The SIMPLE algorithm is employed for pressure–velocity coupling, and convergence criteria are set to ensure solution accuracy. Key flow parameters such as velocity distribution, pressure contours, and streamline patterns are extracted from the simulation.

Finally, a comparative analysis is conducted between experimental observations and CFD results to validate the accuracy of the developed setup. The agreement between flow visualization patterns and numerical predictions confirms the reliability of the methodology and the effectiveness of the water tunnel for aerodynamic analysis.

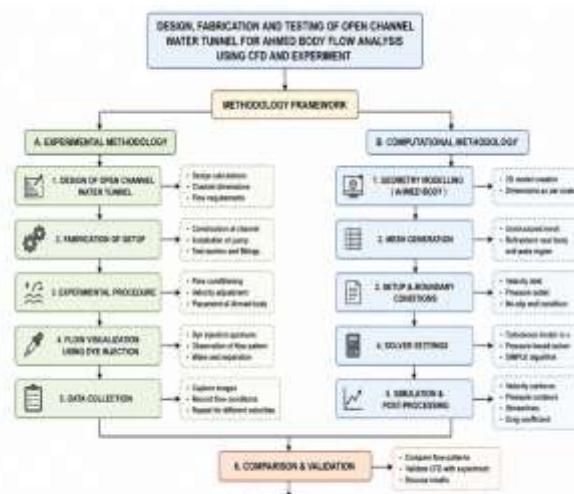


Fig. 1 Block diagram of methodology

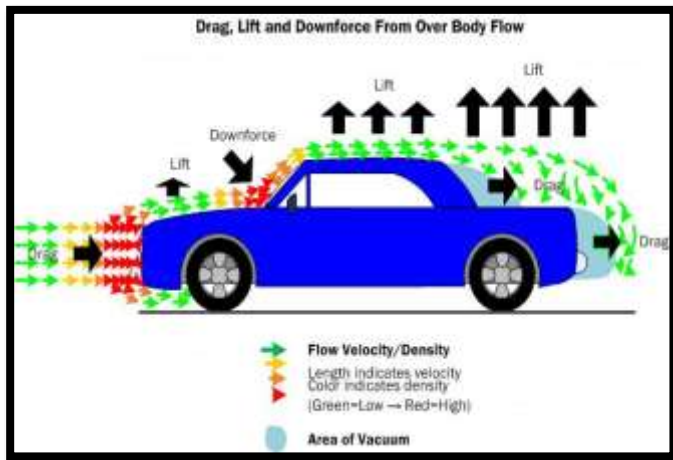
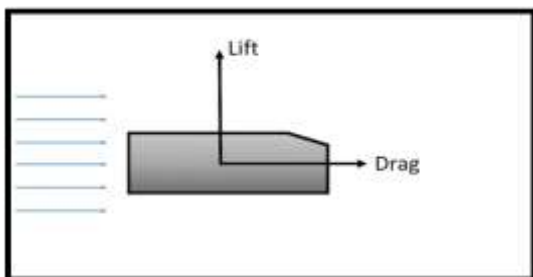


Fig 2: Coefficient of drag

V. Drag & Lift

Aerodynamic Drag



VI. Trails Photos



Fig 4. Trails conducted



Fig 5. Water tunnel and die trail setup

Flow Behavior Around Ahmed Body

Aerodynamic Drag

Aerodynamic drag is the resistance force experienced by a body when it moves through a fluid. In the case of the Ahmed body, drag primarily arises due to pressure differences between the front and rear surfaces, as well as viscous effects along the body surface. When fluid flows over the body, a boundary layer develops, and due to the bluff nature of the geometry, flow separation occurs at the rear slant. This leads to the formation of a low-pressure wake region, which significantly increases pressure drag.

The drag coefficient (C_d) is a dimensionless parameter used to quantify this resistance and is influenced by factors such as body shape, surface roughness, and flow velocity. For bluff bodies like the Ahmed body, the drag coefficient is relatively high due to large wake formation. Reducing drag is essential in automotive design, as it directly impacts fuel efficiency and performance.

Aerodynamic Lift

Aerodynamic lift is the force acting perpendicular to the direction of fluid flow and is generated due to pressure differences between the upper and lower surfaces of a body. In the Ahmed body, lift is not always desirable, as excessive positive lift can reduce vehicle stability, especially at high speeds. The slanted rear geometry influences the pressure distribution, causing variations in lift behavior.

The lift coefficient (C_l) represents the magnitude of lift generated and depends on the flow characteristics and body geometry. In automotive applications, controlling lift is crucial, and often a slight negative lift (downforce) is preferred to improve traction and stability. Understanding lift behavior in the Ahmed body helps in optimizing vehicle design for better aerodynamic performance.

VII. Numerical Investigations

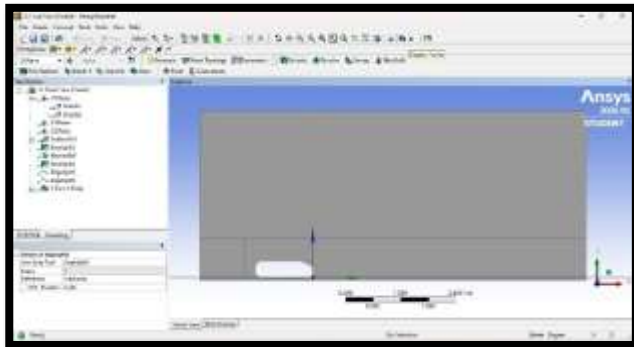


Fig 5. 2D Drawing of a setup in Ansys

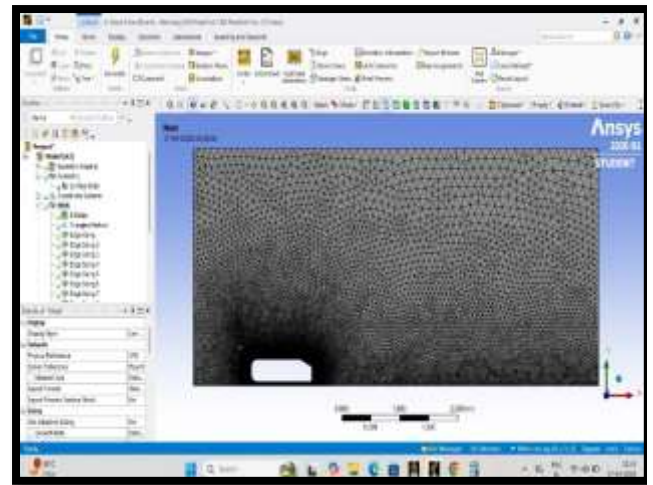


Fig 6. Mesh of a 2D drawing

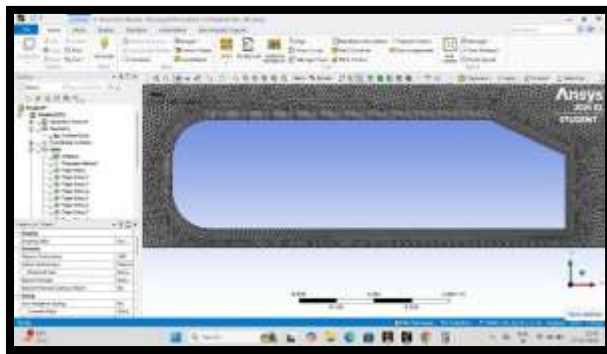


Fig 7. Fine Mesh Near Ahmed Body



Fig 8. computational Setup

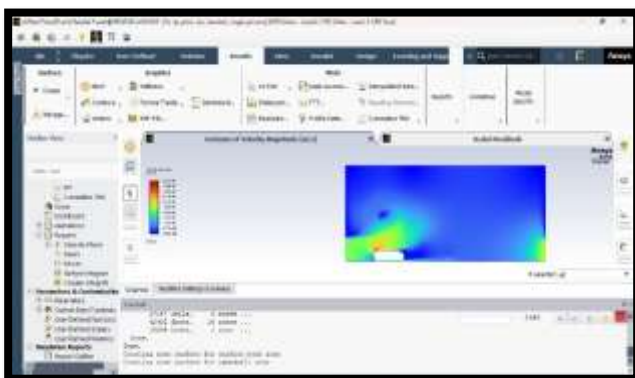


Fig 9: Streamline Contour of Velocity magnitude

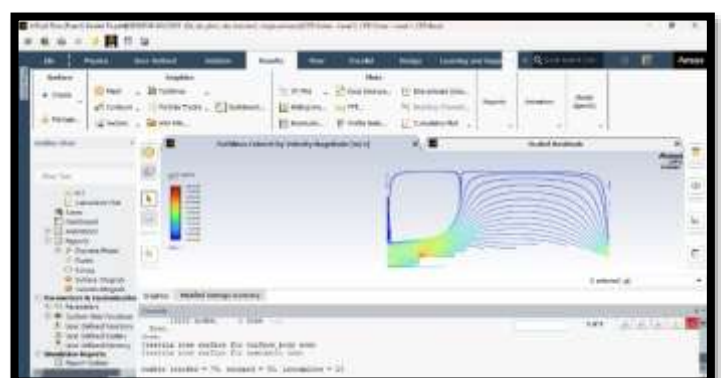
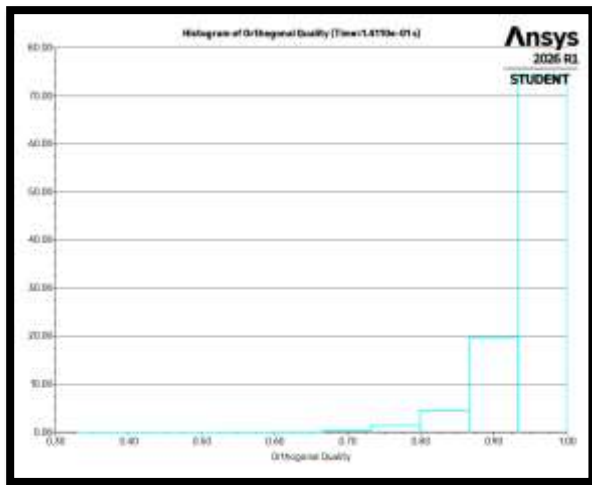


Fig 10. Pathline Contour Of velocity Magnitude

VIII. Results



Area	1m ²
Density	1.225 kg/m ³
Depth	1m
Enthalpy	0 J/kg
Length	1m
Pressure	0 Pa
Temperature	288.16 K
Veloccity	1m/s
Viscosity	1.7894e-05 kg/(m s)
Ratio of Specific Heats	1.4
Yplus for Heat Tran. Coef.	300

Solution Status

	Value	Absolute criteria	Convergence status
Continuity	0.0002181259	0.001	Converged
x-velocity	0.0002764391	0.001	Converged
y-velocity	0.0002871991	0.001	Converged
k	0.0010111102	0.001	Not Converged
epsilon	0.001026482	0.001	Not Converged

IX. CONCLUSION

The study successfully demonstrates the design and development of an open channel water tunnel for aerodynamic analysis.

The experimental results show strong correlation with CFD simulations, validating the effectiveness of the setup. The system provides a cost-effective alternative for aerodynamic testing and enables clear visualization of flow behavior.

Future work can include advanced techniques such as Particle Image Velocimetry (PIV) and improved turbulence modeling..

REFERENCE

- Ahmed, S.R., Ramm, G., Faltin, G. (1984) – SAE Paper
- Franck, G. et al. (2009) – CFD Study
- Khan, R.S., Umale, S. (2014) – Turbulence Models
- Bammidi, R., Murty, B.V. (2014) – ANSYS CFX
- Kumar, G. et al. (2020) – Numerical Analysis
- Östh, J. et al. (2013) – Advanced CFD
- Buscariolo, F. et al. (2020) – High Fidelity CFD
- Ashton, N. et al. (2024) – AhmedML Dataset
- Batay, S. et al. (2024) – Optimization Study