Aero Dynamic Analysis on Audi Car Model Using CFD

[1]A. Praveen Kumar [2], V. Pradeep Kumar,
[1]PG Scholar[1], Avanthi Institute of Engineering and Technology
[2]Asooc. Professor, Avanthi Institute of Engineering and Technology

***Abstract*— The aerodynamics of vehicles is crucial in today's world of increased competition. Vehicle performance is impacted by aerodynamics because of changes in variables like lift and drag forces, which are important for high speed and fuel efficiency. As computer technology advances, manufacturers are turning to computational fluid dynamics rather than wind tunnel testing in an effort to cut costs and testing time. The aim of the work is to improve a vehicle's design in order to lower the aerodynamic drag and lift coefficient. This can be achieved by positioning the scuttle in front, altering the rear bulkhead, and changing the angle between the car body's base and rear underbody. It increases the fuel efficiency and safety.**

Keywords*—CFD Analysis, *Drag, Lift and Problem analysis.

I. INTRODUCTION

In the automotive field, aerodynamics focuses on the dynamics associated with road vehicles. The primary goals are to decrease drag and wind noise, limit noise emissions, and avert unwanted lift forces and other factors that can lead to aerodynamic instability at increased speeds. Here, air is treated as a fluid in this context. For certain types of racing cars, generating down force may also be crucial for enhancing vehicle traction and improving cornering performance.

Aerodynamics began to be applied to enhance the performance of race cars in the 1970s. Engineers in the racing industry discovered that the airflow around vehicles could help improve down force and minimize aerodynamic drag. With an increasing emphasis on fuel efficiency in the design of road vehicles, engineers quickly recognized that techniques used to lower aerodynamic drag in race cars could be adapted for road vehicles to enhance fuel economy. To lessen the drag produced by a vehicle, car manufacturers started to adopt body designs that allowed for improved streamlining Methods for reducing a vehicle's drag coefficient involve modifying the shape of the rear end, enclosing the underside of the vehicles, and minimizing any protrusions on the car's surface.

1.1 Aerodynamics and Fuel Economy

Picture yourself gripping a large traffic cone outside your car window while driving on a freeway at seventy-five miles per hour. You have two choices: either hold the cone with the pointed end directed in the same direction as the car's movement, or with the pointed end facing against the car's movement. In which situation would it be easier to maintain your grip on the cone, considering the impact of air resistance? Although intuition may lead many to believe that pointing the cone forward is more effective, the reality is quite the opposite. This occurrence illustrates aerodynamic drag, a concept that engineers apply to shape a car effectively and reduce the engine power required to propel it forward. By refining a vehicle's design to minimize aerodynamic drag, automotive designers can create cars that achieve better fuel efficiency. The significance of fuel economy for manufactured vehicles is increasingly clear. Enhanced automotive aerodynamics results in lower fuel consumption, which benefits drivers financially and reduces carbon dioxide emissions. One of the critical factors that contemporary automotive engineers consider when designing a vehicle is its aerodynamic properties. Aerodynamics is the study of how air moves and the forces exerted on objects as they move through it. When a car is in motion, it displaces a significant volume of air, which then flows around the vehicle.

II. LITERATURE REVIEW

Aerodynamics is the study of air movement and the forces acting on objects as they move through the air. When a car is in motion, it displaces a significant volume of air, which must flow around it. While the study of aerodynamics has been ongoing since the 19th century, its significance for automobiles became clear only in the mid-20th century. Engineers discovered that adding aerodynamic features to race cars could influence their performance through two key effects: down force and aerodynamic drag.

Down force and aerodynamic drag are two forces generated by the airflow around a car. Down force refers to the downward pressure on a vehicle caused by differences in air pressure, which enhances performance by pushing the car closer to the ground. In contrast, aerodynamic drag is the force that opposes the vehicle's motion. To integrate inverted wings into race cars without significantly increasing induced drag, engineers developed techniques to minimize drag, thus reducing the engine power required to maintain a specific speed. By lowering drag, a car's fuel efficiency can also improve.

Sneh Hetawal et al. [3] 2014, in the article describes the design and CFD analysis of a Formula SAE car. The main goal of this study is to enhance the stability of the vehicle and reduce the drag. With this the track performance will be increased also the resistance of air to the vehicle gets reduced.

S.M. Rakibul Hassan et al. [6]2013in their research concentrated on different aspects analysis of aerodynamic drag of racing cars and different drag reduction techniques such as rear under body modification and exhaust gas redirection towards the rear separation zones. The drag can be reduced up to 22.13% by different rear under-body modifications and up to 9.5% by exhaust gas redirection towards the separated region at the rear of the car.
S.Y. Cheng et al.[8] 2011, in their research the influence of transient flows on vehicle stability was investigated by large eddy simulation. To consider the dynamic response of a vehicle to real-life transient aerodynamics, a dimensionless parameter that quantifies the amount of aerodynamic damping for vehicle subjects to pitching oscillation is proposed. Like so many authors, done their research on car like bodies in aerodynamics
Based on the previous work the objective fixed as follows,

- By increasing angle between the edge of the rear end to the ground which leads to decrease the drag force.
- By using aerofoil structure on front spoiler in design increases stability of the vehicle and decreases the drag force.

III. COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is the study of forecasting fluid flow, heat transfer, mass transfer, chemical reactions (such as combustion), and other related phenomena. This is achieved by solving the mathematical equations that describe these processes through numerical algorithms on a computer. CFD is a highly effective technique with applications across various industrial and non-industrial fields.

3.1 GOVERNING EQUATIONS OF FLUID FLOW

The governing equations of fluid flow are mathematical expressions of the physical conservation laws. Each equation corresponds to a specific conservation principle. The core equations of fluid dynamics are derived from three universal conservation laws: 1) Conservation of mass 2) Conservation of momentum 3) Conservation of energy

3.2 Working of CFD

All CFD codes consist of three primary components: 1. Preprocessor 2. Solver 3. Postprocessor

3.3.1Pre Processor

The process involves transferring the input for a flow problem to a CFD program through a user-friendly interface, followed by converting this input into a format suitable for the solver. The pre-processing activities are carried out in stages and include: 1. Defining the geometry of the area of interest, known as the computational domain. 2. Generating a grid or mesh, which subdivides the computational domain into smaller segments called cells or control volumes. 3. Selecting the physical and chemical phenomena that need to be modeled. 4. Defining the properties of the fluid. 5. Specifying the appropriate boundary conditions for the cells that are adjacent to the domain boundary.

The solution to a flow problem—such as pressure, velocity, or temperature—is determined at 'nodes,' which are located at the corners of each cell. The accuracy of a CFD solution is influenced by the number of cells in the grid; generally, a higher number of cells leads to greater accuracy. However, this also results in longer computation times.

3.3.2 Solver

There are three main numerical solution techniques: finite difference, finite element, and finite volume methods. The numerical methods that underpin the solver follow these key steps: 1. Approximate the unknown flow variables using simple functions. 2.

Discrete the problem by substituting the approximations into the governing equations and performing mathematical manipulations. 3. Solve the resulting algebraic equations through an iterative process.

3.2.3. Post Processor

Similar to preprocessing, there has been significant development in the field of post-processing recently. With the growing popularity of engineering workstations, many of which boast exceptional graphics capabilities, leading CFD packages now come with a range of versatile data visualization tools. These tools include: 1) Domain geometry and grid display 2) Vector plots 3) Line and shaded contour plots 4) 2D and 3D surface plots 5) Particle tracking 6) View manipulation 7) Postscript

3.3.3 Finite Volume Solver

FLUENT is an advanced computer program designed for modeling fluid flow and heat transfer in intricate geometries. It offers complete mesh flexibility and can effectively solve flow problems using unstructured meshes that are easily generated around complex shapes. The program supports various mesh types, including 2D triangular and quadrilateral, as well as 3D tetrahedral, hexahedral, pyramid, wedge, and hybrid meshes. Additionally, FLUENT features grid adaptation, allowing users to refine or coarsen the grid based on the flow solution. This adaptive grid capability is especially beneficial for accurately predicting flow fields in areas with significant gradients, such as free shear layers and boundary layers. Compared to solutions on structured or block-structured grids, this feature greatly reduces the time needed to create a high-quality grid. Solution-adaptive refinement simplifies grid refinement studies and minimizes the computational resources needed to reach a specific level of accuracy by focusing mesh refinement only in areas that require higher resolution. The fundamental steps in this process are as follows: 1. Define objectives. 2. Create the model geometry and grid. 3. Select the solver and physical models. 4. Compute and monitor the solution. 5. Review and save the results.

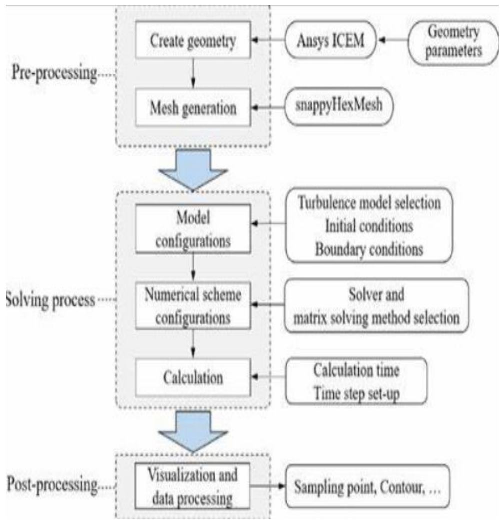

Fig 3.1; Flow chart showing complete CFD analysis

IV. DESIGN AND ANALYSIS

A.EXISTING MODEL

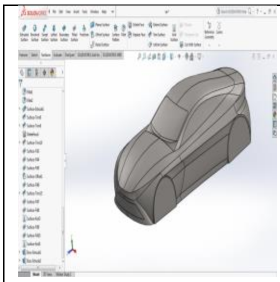	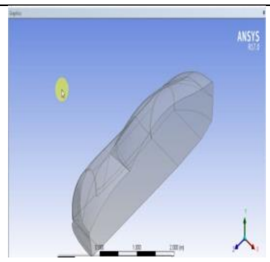
Fig 4.1: Geometry of Audi r8 existing car profile	Fig 4.2: Geometry in Design Modeler
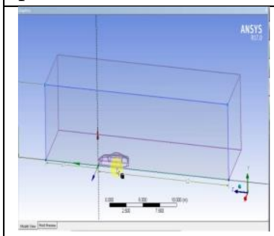	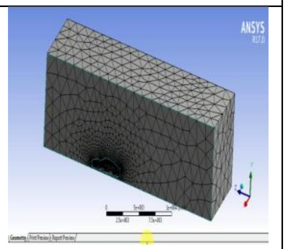
Figure 4.3: Domain of the Geometry	Figure 4.4: Meshing of car profile
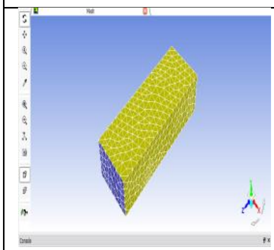	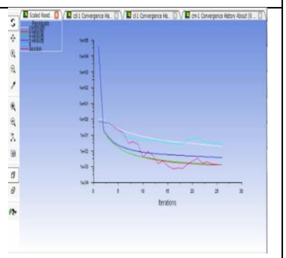
Figure 4.5: Boundaries of the profile	Fig 4.6: Simulation graphs

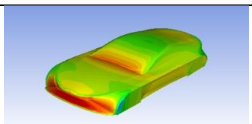	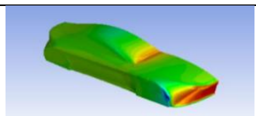
Fig 4.7.1 Contours of absolute pressure@80 Km/hr	Fig 4.8.1 Contours of absolute pressure@100 Km/hr
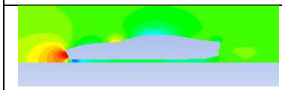	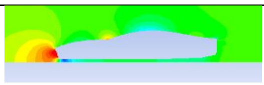
Fig 4.7.2 Contours of static pressure@80 Km/hr	Fig 4.8.2Contours of static pressure@100 Km/hr
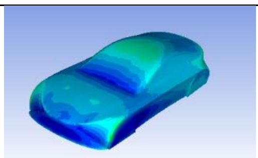	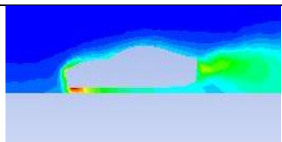
Fig 4.7.3 Contours of dynamic pressure@80 Km/hr	Fig 4.8.3 Contours of dynamic pressure@100 Km/hr
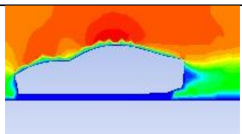	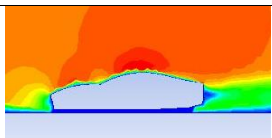
Fig 4.7.4 Contours of velocity magnitude@80 Km/hr	Fig 4.8.4 Contours of velocity magnitude@100 Km/hr

4.1 SIMULATION RESULTS AND POST-PROCESSING:

Table 4.1: Results for Audi r8

S. No	Velocity of the car in (km/hr)	Drag-coefficient (c_d)	Lift coefficient (c_l)
1	80	0.348	0.43
2	110	0.357	0.464

The above table 4.1 shows the values of pressure distribution and drag force over the surface of the profile of the car for given speeds (80, 110 km/hr) which are obtained from the post-processing results in CFD-POST.

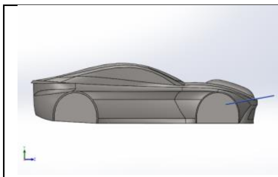	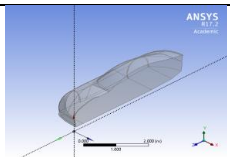
Fig4B.1: Geometry in Design Modeler	Fig 4B.2: Geometry imported to work bench Design Modeler

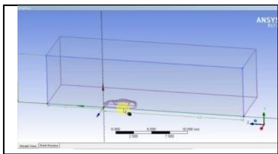	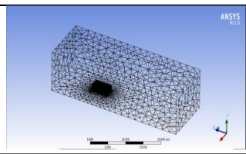
Fig 4B.3: Domain of the Geometry	Fig4B.4: Meshing of car profile
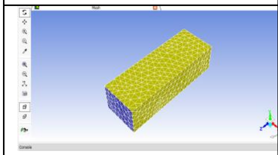	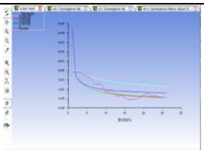
Fig4B.5: Boundaries of the profile	Fig 4B.6Simulation graphs
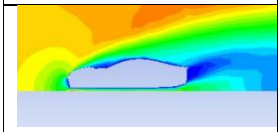	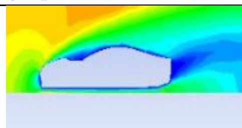
Fig4B.7.1 Contours of static pressure@80 Km/hr	Fig 4B.8.12Contours of static pressure@100 Km/hr
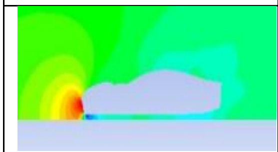	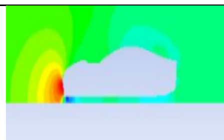
Fig4B.7.2 Contours of dynamic pressure@80 Km/hr	Fig 4B.8.2 Contours of dynamic pressure@100 Km/hr
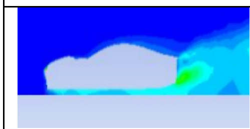	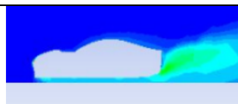
Fig4B.7.3 Contours of velocity magnitude@80 Km/hr	Fig4B.8.3 Contours of velocity magnitude@100 Km/hr

B.MODIFIED MODEL

4.2 SIMULATION RESULTS AND POST-PROCESSING:

Table 4.2: Results for Audi r8 modified car profile

S. No	Velocity of the car in (km/hr)	Drag coefficient (c_d)	Lift coefficient (c_l)
1	80	0.224	0.277
2	110	0.311	0.443

4.3 CHECK FOR BOUNDARY LAYER SEPARATION:

BOUNDARY LAYER SEPARATION CHECK FOR EXISTING CAR BODY:

Boundary layer separation exists when there is adverse pressure gradient (∂P/∂x) <0. From the pressure contour, (∂P/∂x) is greater than zero.

BOUNDARY LAYER SEPARATION CHECK FOR MODIFIED CAR BODY:

Boundary layer separation exists when there is adverse pressure gradient ($\partial P/\partial x$) <0. From the pressure contour ($\partial P/\partial x$) is greater than zero

V. RESULTS

5.1 Comparison results between existing and modified car body drag coefficient

	Speed (km/hr)	Existing car body (c_d)	Modified car body (c_d)	(%)of drag reduction
1	80	0.348	0.224	35.63
2	100	0.357	0.311	12.88

5.2 comparison results between existing and modified car body lift coefficient

S.No	Speed (km/hr)	Existing car body (c_l)	Modified car body (c_l)	(%)of lift reduction
1	80	0.43	0.277	35.33
2	100	0.464	0.443	4.52

VI. CONCLUSIONS & FUTURE SCOPE

6.1 CONCLUSION

Simulation results indicate that the modified car design exhibits lower drag and reduced average static pressure compared to the existing Audi R8 design at the same angle and speed. The minimum drag was recorded following the enhancements made to the front and rear of the car body. Consequently, our design experiences less turbulence, which contributes to improve fuel efficiency and enhanced vehicle stability.

6.2 FUTURE SCOPE

This type of testing can be carried out for various profiles at various air flow velocities and orientations. Through the use of wind tunnel tests with varying test section sizes, average static pressure and drag force can be determined by varying the air flow velocity for a specific profile at different designs. This analysis may be conducted for any type of four-wheeler car profile, which will help us boost vehicle stability and lessen the impact of drag.

REFERENCES

(Periodical style)

[1] James Keog, etal. The influence of cornering on the vortical wake structures of an inverted wing, volume 229, Issue 13, 2016.

[2] Alamaan Altaf a, et al. Passive drag reduction of square back road vehicles Volume 134, November 2014, Pages 30-43, 2014.

[3] Sneh Hetawala, et al. Aerodynamic Study of Formula SAE Car Volume 97, 2014, Pages 1198-1207, 2014.

[4] D.E. Aljure a , et al. Flow and Turbulent Structures Around Simplified Car Models), 2014.

[5] J.H. Amorim, et al. CFD modelling of the aerodynamic effect of trees on urban air pollution dispersion, 2013.

[6] S.M. Rakibul Hassan, et al. Numerical Study on Aerodynamic Drag Reduction of Racing Cars, Volume 90, 2014, Pages 308-313, 2013.

[7] R. P. Littlewood M. et al. Aerodynamic drag reduction of a simplified square back vehicle using steady blowing Volume 53, pages 519-529, 2012

[8] S.Y. Cheng a et al. A numerical analysis of transient flow past road vehicles subjected to pitching oscillation Volume 99, Issue 5, Pages 511-522, 2011.

[9] Chien-Hsiung Tsai a et al. Computational aero-acoustic analysis of a passenger car with a rear spoiler, Volume 33, Issue 9, September 2009, Pages 3661-3673, 2009.

[10] W. Angelis et al. Numerical and experimental study of the flow over a two-dimensional car model, Volume 62, Issue 1, Pages 57-79, August 1996