The Impact and Analysis of Vortex Generators on Car Aerodynamics

Anshool Naik[1], Ayush Patil[2],
Atharva Pawar[3], Pranav Pilankar[4],
Prabodh Horambe[5] Assistant Professor [5],
Department of Mechanical Engineering,
Finolex Academy of Management and Technology, Ratnagiri, India

---***---

Abstract- *Aerodynamic performance plays a crucial role in modern automobile engineering, particularly in enhancing vehicle stability, fuel efficiency, and high-speed performance. The presence of drag and lift forces significantly affects vehicle dynamics, especially at higher speeds. This research investigates the application of vortex generators on a car's surface to minimize drag and lift forces. The study utilizes Computational Fluid Dynamics (CFD) simulations to analyze airflow behavior. The results indicate that the implementation of appropriately designed vortex generators leads to a notable reduction in drag force and improved vehicle stability.*

Keywords- *Aerodynamics, Vortex Generators, Drag Reduction, Lift Force, CFD Analysis*

I. INTRODUCTION

In recent years, the automotive industry has witnessed increased emphasis on optimizing vehicle aerodynamics to enhance performance, safety, and fuel economy. Aerodynamic forces such as drag and lift directly influence vehicle speed, handling, and energy consumption. Various aerodynamic accessories, including spoilers, diffusers, and vortex generators, are incorporated to improve airflow behavior around the vehicle.

Vortex generators are small, fin-like components strategically positioned on a vehicle's surface to control airflow separation and induce longitudinal vortices. These vortices energize the boundary layer and delay flow separation, thereby reducing drag and lift forces. The objective of this project is to evaluate the impact of different vortex generator designs on a car's aerodynamic characteristics and determine the most effective configuration.

II. LITERATURE REVIEW

Pravin B. Pachpor et al. (IJERT, 2015) [1]
An attempt was made to analyze the aerodynamic behavior of a notchback car geometry using vortex generators. The research involved experimental and computational analysis to evaluate drag reduction by delaying flow separation. It was observed that the introduction of trapezoidal-shaped vortex generators at an optimized location led to a reduction in the drag coefficient. The study concluded that the vortex generators helped energize the boundary layer and controlled the separation of airflow at the rear windshield, ultimately reducing drag forces acting on the vehicle.

S. Vishal et al. (ICMERE, 2015) [2]
This research focused on the CFD analysis of flow separation control over a sedan car model using vortex generators. Their simulation results demonstrated that the addition of vortex generators on the roof region of the sedan delayed the boundary layer separation, minimized wake formation, and improved airflow reattachment. The drag coefficient was significantly reduced as compared to the baseline model. The study also emphasized the importance of the vortex generator's shape, size, and orientation in achieving optimal aerodynamic performance.

P. J. Rao et al. (IJSR, 2013) [3]
This studied various passive flow control devices were evaluated for their effect on reducing aerodynamic drag. The research highlighted the contribution of devices like diffusers, spoilers, and vortex generators in enhancing vehicle aerodynamics. Specific emphasis was placed on the role of vortex generators in creating longitudinal vortices that re-energized the boundary layer. The results validated that proper placement of vortex generators successfully delayed flow separation, leading to a considerable reduction in pressure drag and improvement in vehicle stability.

III. METHODOLOGY

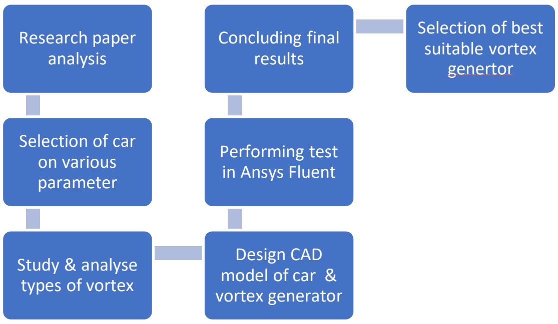


Fig 1 Work flow chart

CAD models of six vortex generator designs were developed and simulated in ANSYS Fluent. A simplified car model was also created for full-system analysis. Laminar, steady-state simulations were performed at 50 m/s inlet velocity. Mesh generation used tetrahedral elements to accommodate VG.

Developing CAD models of Vortex Generators

The vortex generators were designed using Fusion360 and SolidWorks CAD softwares to accurately represent various geometric configurations intended to influence airflow over the vehicle body. Six distinct vortex generator shapes were modelled. The images of the designed vortex generator models are included below:

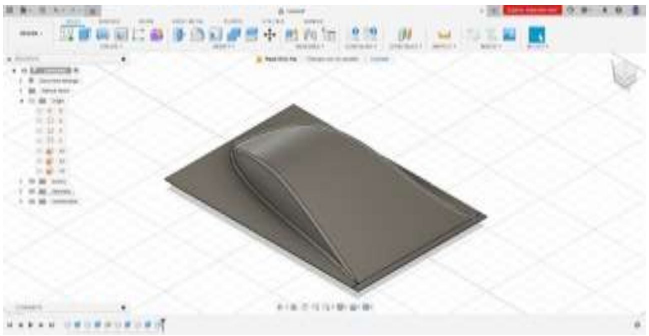


Fig 1.1 Vortex type 1

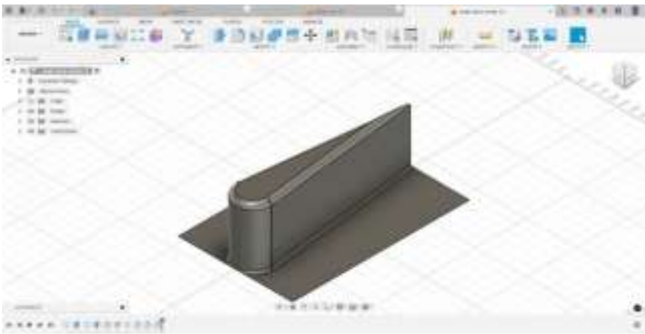


Fig 1.2 Vortex type 2

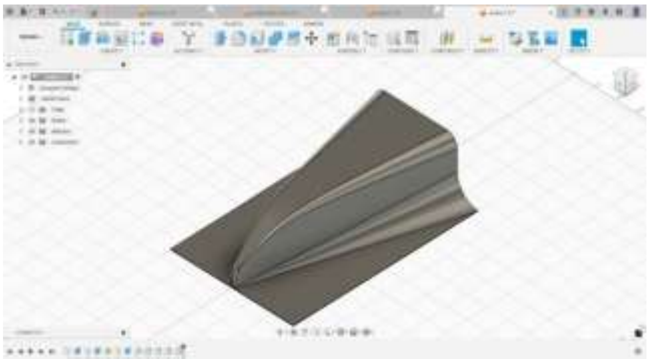


Fig 1.3 Vortex type 3 (modified)

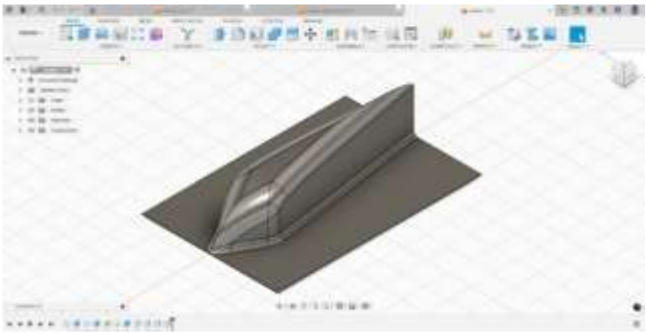


Fig 1.4 Vortex type 4

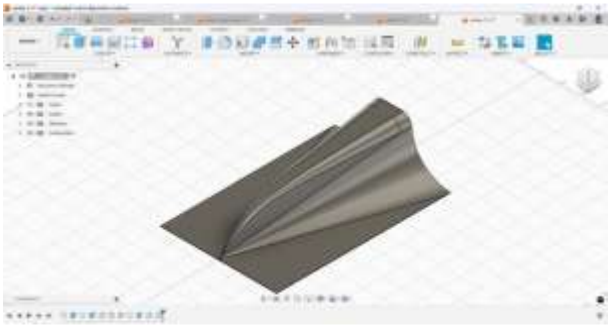


Fig 1.5 Vortex type 5

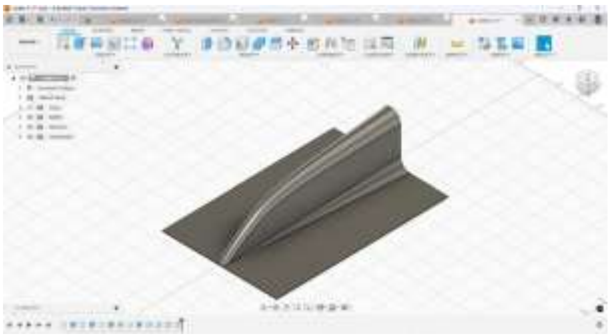


Fig 1.6 Vortex type 6

Creating CAD model of the car

A simplified car body was designed using Fusion360 to serve as the base for aerodynamic analysis. The model includes a smooth front surface, flat sides, and a sloped rear to simulate real-world airflow behaviour. Unnecessary details were excluded to reduce complexity and focus on the wake region.

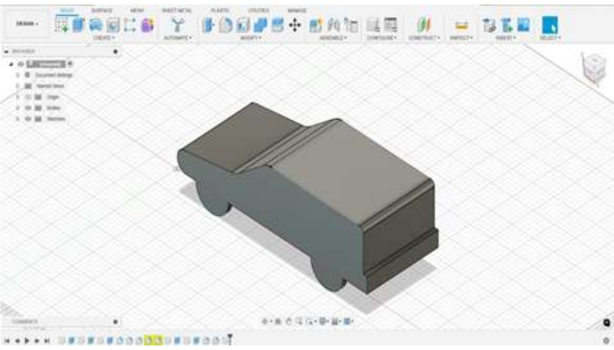


Fig 1.7 CAD MODEL OF CAR

Conducting performance analysis of each vortex in ANSYS Fluent

Performance analysis of the different vortex generator designs was conducted using ANSYS Fluent, a powerful tool for simulating fluid flow and analyzing aerodynamic behavior. Each vortex generator model was evaluated for its impact on airflow characteristics such streamline and pathline velocities. The simulation environment was set up by launching ANSYS Fluent through the ANSYS Workbench platform.

ANSYS Workbench Layout

The ANSYS Workbench Project Schematic provides a visual layout for managing the simulation workflow. It displays the different stages such as geometry import, meshing, setup, solution, and results in a connected flow. Each block represents a step, and their connections show the data transfer sequence.

Fig 2 ANSYS window layout

Importing Geometry

The CAD models ox vortex generators were imported into ANSYS Fluent.

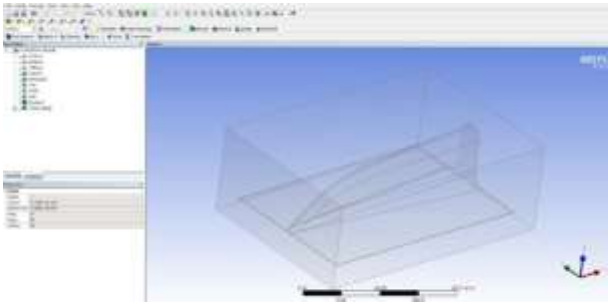


Fig 2.1 Imported Geometry of vortex type 6

Meshing the Geometry

A detailed tetrahedral mesh was generated of each vortex. A tetrahedral mesh was chosen for this project because it easily adapts to complex geometries vortex generators. It allows better coverage of curved surfaces and small features without requiring complicated manual meshing. Tetrahedral elements provide a good balance between accuracy and computational efficiency, making them ideal for aerodynamic simulations involving detailed 3D models.

Enclosure space: 30x30mm

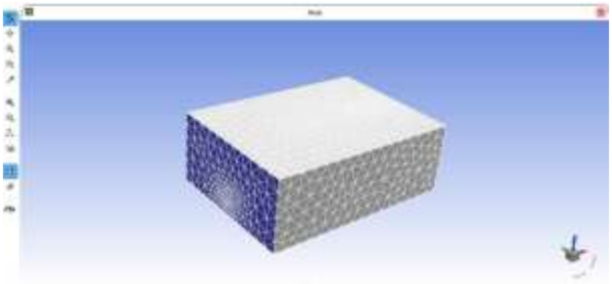
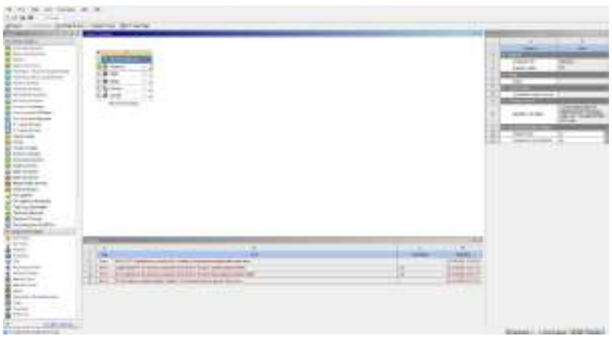


Fig 2.2 Meshed geometry of vortex type 6

Defining Boundary Conditions

Flow type: Fluid- Air, Steady, laminar airflow. **Boundary condition**: Magnitude, normal to **boundary Inlet boundary**: Velocity inlet at 50 m/s.
Enclosure space: 30x30mm



Body material – Aluminium

Fig 2.2 Boundary Condition of vortex type 6

Solution initialization and running the simulation

The flow field was initialized from the inlet. 200 iterations were performed until result was gained.

Post-Processing the results

Velocity streamlines and pathlines were analyzed to observe flow reattachment and wake behaviour. Also scaled iterations graph was extracted.

The above procedure was followed for all vortex generator analysis.

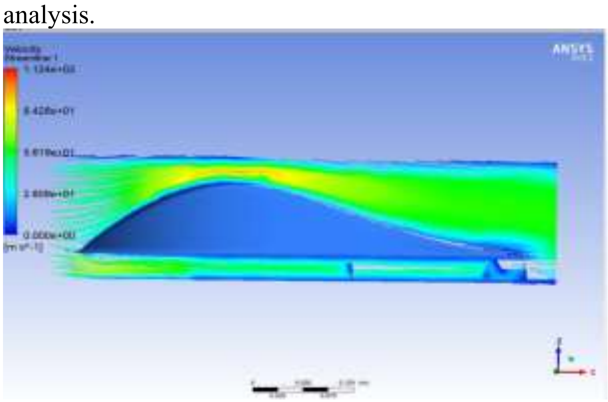


Fig 3.Streamline velocity analysis of vortex type 1

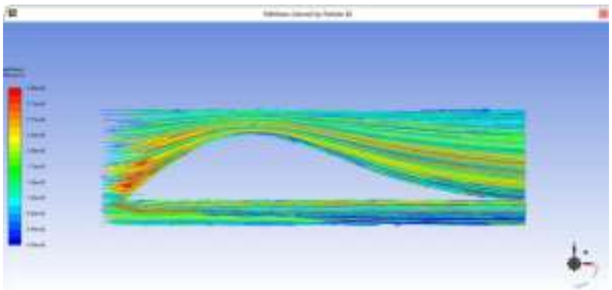


Fig 3.1 Pathline velocity analysis of vortex type 1

The initial version of the vortex generator type 1 was designed and prepared for simulation. However, during early analysis, it failed to deliver consistent and realistic aerodynamic behaviour. Issues such as sharp edges, poor surface continuity, and ineffective vortex formation were identified, leading to inaccurate results.

In response, a detailed review of the design principles was conducted with a student- level understanding of fluid flow and vortex dynamics. Based on these insights, an updated version of the vortex generator was developed with significant improvements as type 1.2:

Fig 3.1 Vortex type 1.2

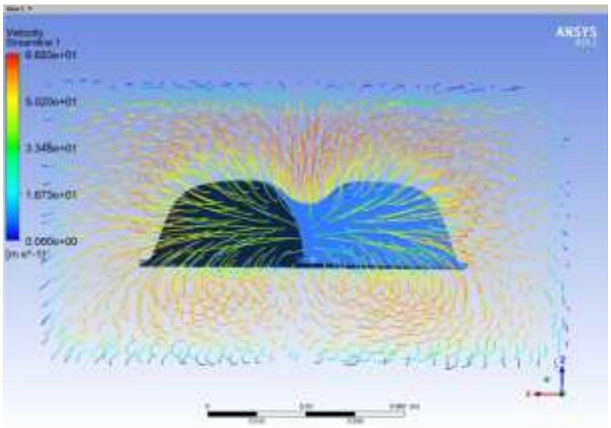


Fig 3.2 Streamline velocity analysis of vortex type 1.2

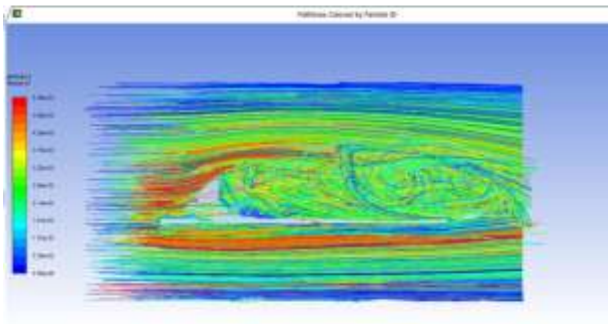


Fig 3.3 Pathline velocity analysis of vortex type 1.2

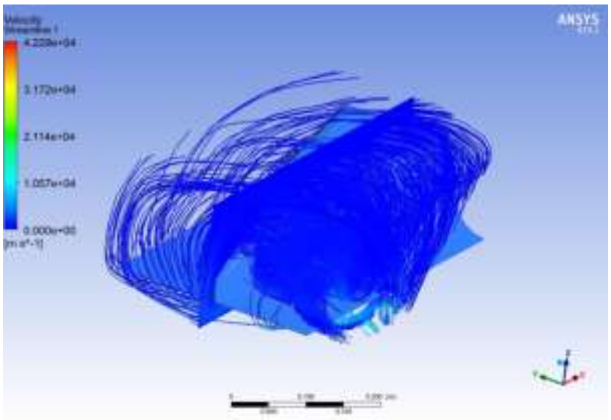


Fig 3.4 Streamline velocity analysis of vortex type 3

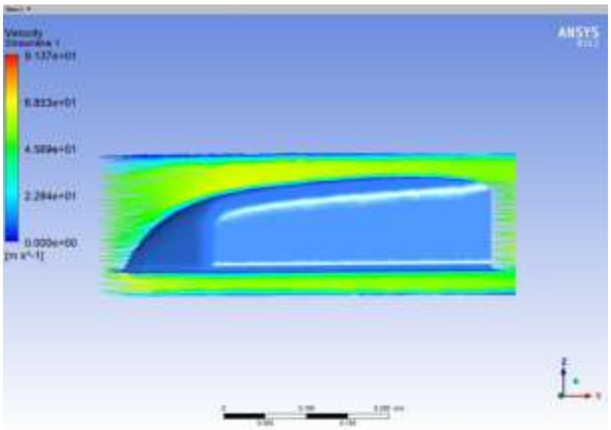


Fig 3.5 Streamline velocity analysis of vortex type

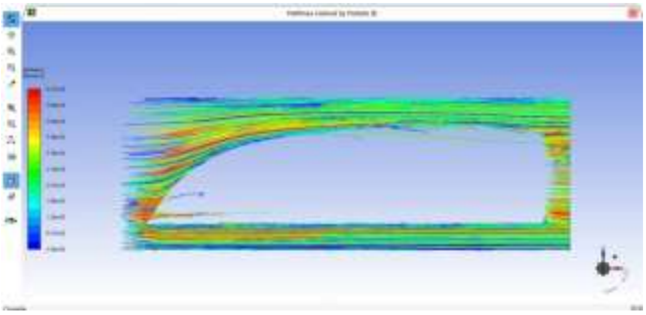


Fig 3.6 Pathline velocity analysis of vortex type 4

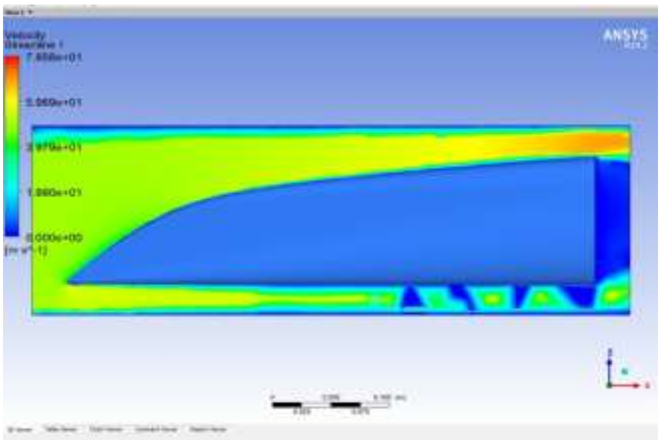


Fig 3.7 Streamline velocity analysis of vortex type 5

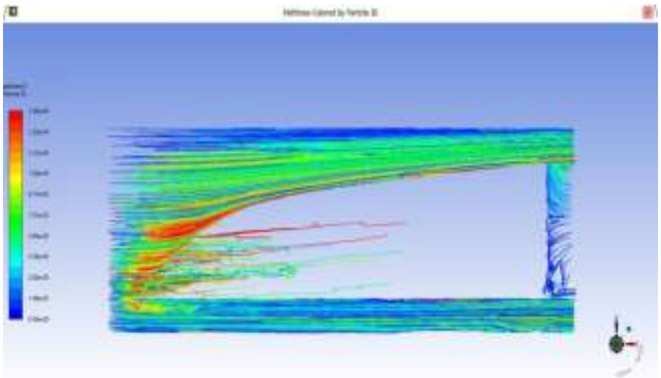


Fig 3.8 Pathline velocity analysis of vortex type 5

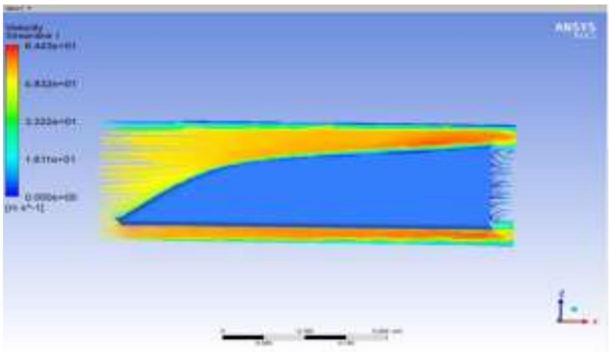


Fig 3.9 Streamline velocity analysis of vortex type 6

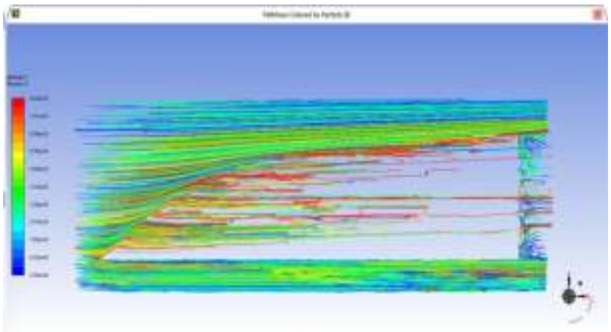


Fig 3.10 Pathline velocity analysis of vortex type 6

Selecting the best performing vortex generator

After designing and simulating the six vortex generators types each was analyzed based on its effect on aerodynamic drag and flow behaviour. CFD simulations were conducted under identical conditions to ensure fair comparison. The obtained residual plots for each vortex are as follows:

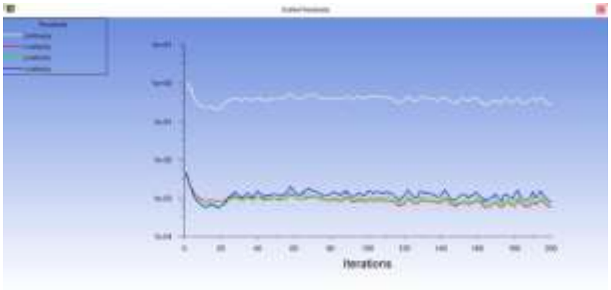


Fig 4 Residual plot of vortex type 1

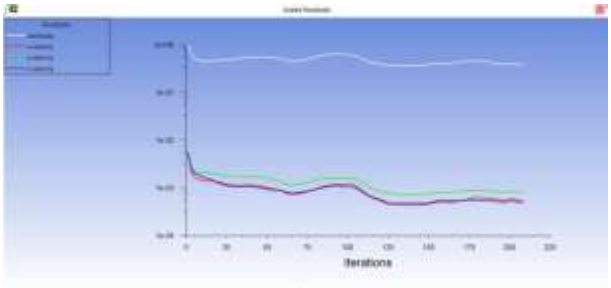


Fig 4.1 Residual plot of vortex type 1.2

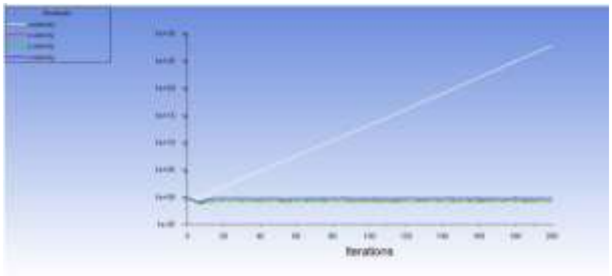


Fig 4.2 Residual plot of vortex type 2

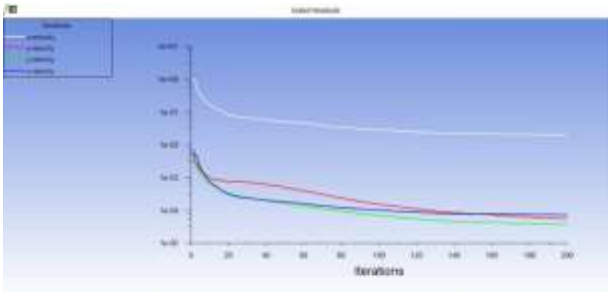


Fig 4.3 Residual plot of vortex type 4

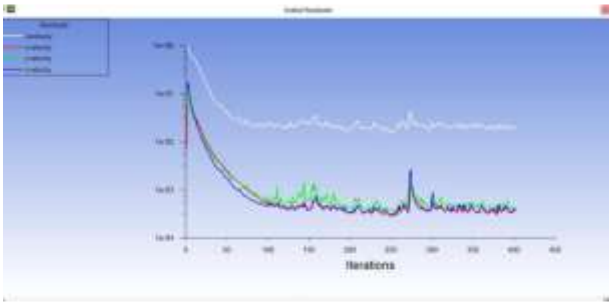


Fig 4.4 Residual plot of vortex type 5

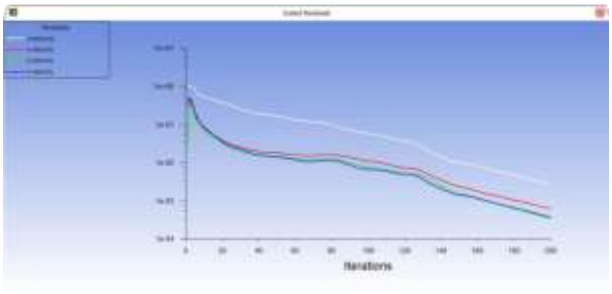


Fig 4.5 Residual plot of vortex type 6

The selection of the best performing vortex generator was made after analyzing simulation results from ANSYS Fluent and model 6 was chosen for further analysis. Various parameters such as flow behaviour, wake structure, and velocity profiles were used to judge the effectiveness of each design.

Basis for Selection of Model 6:

• Maximum Wake Energy Dissipation:

Model 6 demonstrated the highest level of velocity energy loss in the wake region, indicating that the vortex generators strongly influenced the flow and disrupted the large separation zone.

• Wake Reorganization:

Although the velocity dropped significantly behind the body, the flow reattachment downstream was more controlled, leading to better aerodynamic stability.

• Stable Residual Behaviour:

The simulation residuals showed acceptable convergence, suggesting the solution was reliable for interpreting aerodynamic trends.

Model 6 was selected as the best performing vortex generator because it created strong flow disturbances that helped in reducing the adverse pressure in the wake, effectively lowering the overall aerodynamic drag on the vehicle.

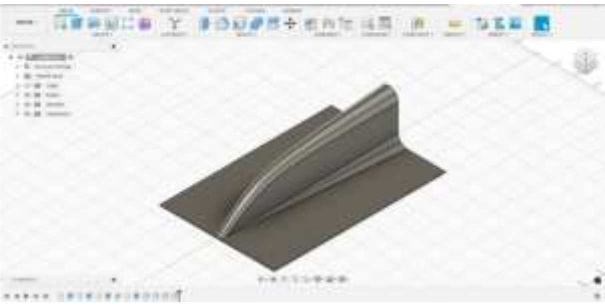


Fig.5 Selected model: Vortex type 6

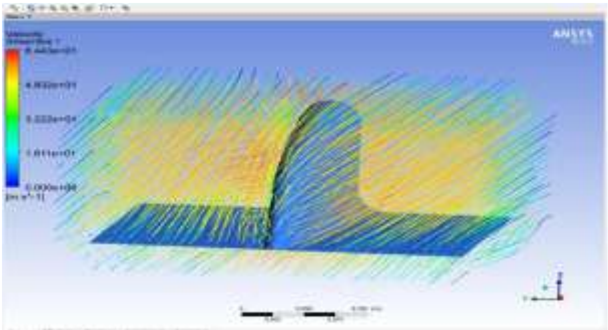


Fig 5.1 Streamline velocity analysis 2 of vortex type 6

Conducting test of selected vortex model with car

CAD model of car with vortex generator type 6

The CAD model of the car with vortex generator type 6 was created by integrating the final optimized VG design onto the rear roof surface of the vehicle. The model was designed in Fusion360 and exported for further meshing and CFD simulation.

Fig 5.2 CAD model of car with vortex generator type 6

Importing geometry to workbench design modeler and meshing

The CAD model of the car with vortex generator type 6 was imported into ANSYS Workbench. Using Design Modeler, the geometry was checked to ensure it was clean and ready for meshing. A fluid domain (air volume) was created around the car model, this allowed the airflow around the vehicle to be properly simulated. A fine mesh was created around important areas like the vortex generators and the car body to capture detailed flow behaviour. Tetrahedral elements were used because they fit well around complex shapes.

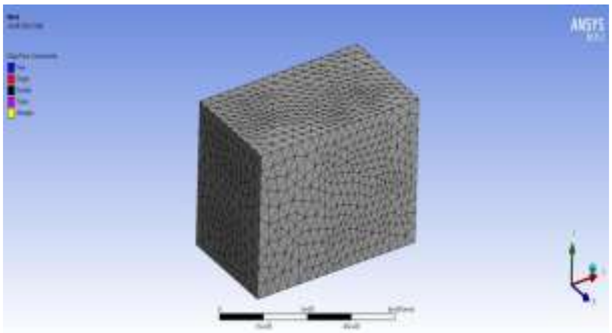


Fig 5.3 Meshing of the domain created and model

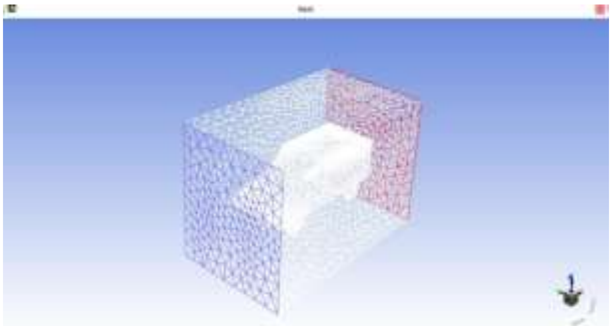


Fig 5.4 Classic mesh view

Solver settings

The CFD simulation setup was completed in ANSYS Fluent after importing the mesh. Solver configuration:

- Solver Type: Laminar, steady-state flow
- Material: Air Boundary conditions:
- Velocity Inlet: 50 m/s
- Walls: No-slip boundary condition.

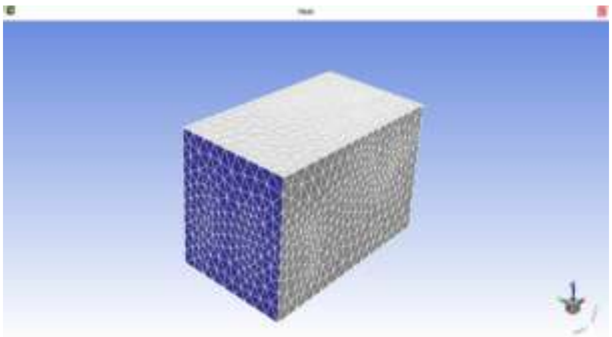


Fig 5.5 Applying boundary conditions

Properties of Schematic A5: Solution

	A	B
1	Property	Value
2	General	
3	Component ID	Solution
4	Directory Name	FFF
5	Use Setup Launcher Settings	✓
6	Precision	Double P...
7	Show Launcher at Startup	✓
8	Display Mesh After Reading	✓
9	Embed Graphics Windows	✓
10	Use Workbench Color Scheme	✓
11	Load ACT Start Page	
12	Environment Path	
13	Setup Compilation Environment for UDF	✓
14	Use Job Scheduler	
15	Run Parallel Version	
16	UDF Compilation Script Path	$(FLUENT_ROOT)\$(ARCH)\udf.bat
17	Initialization Method	Program...
18	Solution Monitoring	✓
19	Generate Solution Monitor Plots for Report	
20	Data Interpolation	✓
21	Notes	
22	Notes	
23	Used Licenses	
24	Last Update Used Licenses	
25	Solution Process	
26	Update Option	Run in F...

Fig 5.6 Solver settings

Solution and post processing

The simulation was run for 200 iterations until all residuals for continuity, momentum, and turbulence equations reached a stable value and plotted smoothly. The residual plots were closely monitored to ensure proper convergence of the solution.

After convergence, velocity streamlines were generated to study the flow behavior around the car and vortex generators. These streamlines helped to visualize how the airflow moved over the vehicle, how the wake region formed behind the car, and how the vortex generators influenced flow separation.

Result

The project titled "The Impact and Analysis of vortex Generator on Car Aerodynamics" gave very positive results in improving the airflow around the car. By adding vortex generators, the airflow stayed attached to the car's surface for a longer time, reducing the size of the wake region behind the car. The CFD simulation showed that the vortex generators helped control the airflow properly and made the pressure at the rear of the car better. The residuals from the simulation also showed that the results became stable after enough iterations. Overall, the use of vortex generators helped in reducing drag, which can improve the car's fuel efficiency and stability, proving that this method is useful for making cars more aerodynamics.

Visual results from the simulation showed that the airflow behind the car was smoother and more controlled compared to normal cases without vortex generators. Overall, this method proved to be effective in improving vehicle performance by potentially increasing fuel efficiency, reducing drag force, and improving stability at higher speeds. These results show that vortex generators can be a simple but powerful addition in the aerodynamic design of vehicles.

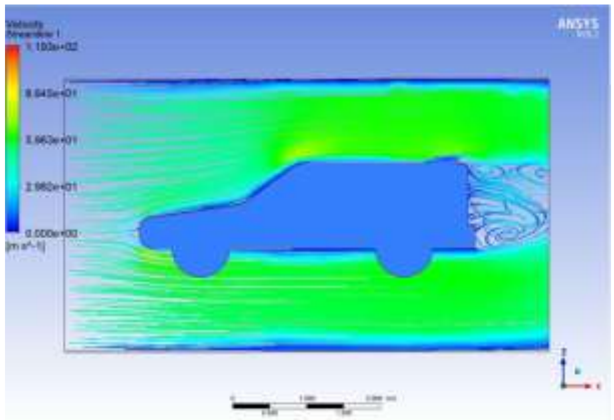


Fig 5.6 Streamline velocity of car with vortex generator

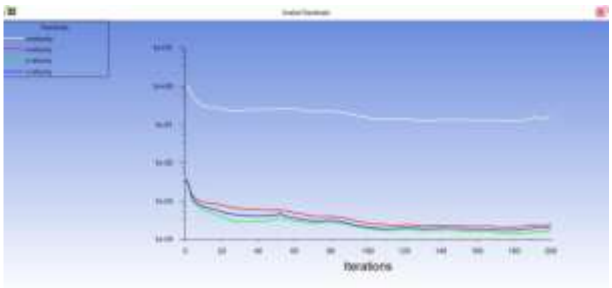


Fig 5.7 Residual plot of car with vortex generator

Residuals analysis

The convergence history shows the scaled residuals for continuity, x-velocity, y- velocity, and z-velocity over 200 iterations.

- Continuity residual started around 100 and reduced by approximately one order of magnitude but did not fall below 10-2. It stabilized after around 80 iterations.
- Velocity residuals (x, y, z components) dropped significantly to values around 10-3 to 10-4, indicating a fairly good convergence for momentum equations.

Effects Observed:

- Improved surface pressure recovery at the rear parts of the car.
- Reduction in size of the wake region behind the car, leading to a decrease in form drag.

The solver reached a semi-converged state suitable for flow analysis, with stable velocity fields. Continuity residual could improve with further mesh refinement or under-relaxation factor tuning, but for preliminary analysis, the results are acceptable.

Conclusion

The project titled "The Impact and Analysis of Vortex Generator on Car Aerodynamics" successfully analyzed the impact of vortex generators on car aerodynamics using CFD simulation in ANSYS Fluent. Different vortex generator designs were studied, and after a detailed comparison based on flow behaviour, wake reduction, and drag performance, vortex generator type 6 was selected as the best performer.

The use of vortex generators helped in controlling the flow separation at the rear of the car, reducing the wake size, and improving overall aerodynamic efficiency. Key outcomes of the project:

- Significant reduction in wake region behind the vehicle.
- Better reattachment of flow and smoother streamline behaviour.
- Improved aerodynamic stability and potential reduction in drag.

The study demonstrated that even small devices like vortex generators can make a noticeable difference in enhancing vehicle performance without major design changes.

Future Scope

While the project achieved its primary objectives, several areas for future improvement and research are identified:

- Wind Tunnel Testing: Physical validation of the CFD results through experimental testing.
- Extended Flow Conditions: Simulations can be performed under various speeds, angles of attack, and crosswinds to study real-world behaviour.
- Real-Time Flow Control: In the future, adaptive vortex generators controlled by AI could dynamically adjust their angles or positions based on real-time driving conditions to maintain aerodynamic efficiency.

By combining advanced CFD methods with AI-based optimization and real-world testing, future research can lead to the development of highly efficient, intelligent aerodynamic systems for next-generation vehicles.

References

[1] Shivam, Sidharth Singh, Abhishek Guleria, 2015, "Aerodynamic Drag Reduction of a Notchback Car Geometry by Delaying Flow Separation using Vortex Generators"

[2] S. Pal, S. M. H. Kabir and M. M. M. Talukder, 201, "Aerodynamic Analysis Of A Concept Car Model"

[3] Gavin Dias, Nisha R. Tiwari, Joju John Varghese, Graham Koyeerath, 2016, "Aerodynamic Analysis of a Car for Reducing Drag Force"