International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 10 | October - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

STUDY OF AERODYNAMIC WAKE ANALYSIS OF SUV CARS FOR IMPROVING FUEL CONSUMPTION

Dr. Sivaraj G[1], Mr. Vignesh A[2], Mr. Abhishek B S[3], Mr. Athineshlal R[4]

[1] Assistant Professor, Aeronautical Engineering & Bannari Amman Institute of Technology
[2] UG Scholar, Automobile Engineering & Bannari Amman Institute of Technology
[3] UG Scholar, Automobile Engineering & Bannari Amman Institute of Technology
[4] UG Scholar, Automobile Engineering & Bannari Amman Institute of Technology

****

ABSTRACT:

SUV cars are known to be some of the most fuel-inefficient vehicles on the road. This is due in part to their large size and shape, which create a lot of aerodynamic drag. The wake region behind an SUV car is particularly turbulent, which further increases drag. The aim of this study of aerodynamic wake analysis of SUV cars for improving fuel consumption is to identify and quantify the factors that contribute to aerodynamic drag in SUV cars, and to develop strategies for reducing drag and improving fuel economy. Identify the factors that contribute to aerodynamic drag in SUV cars. This will involve understanding the flow dynamics of the wake region behind SUV cars and identifying the key features of the wake that contribute to drag. Develop strategies for reducing the size and strength of the wake. This will involve developing methods for modifying the shape of the SUV car or the flow around it in order to reduce drag. Assess the impact of wake reduction on fuel economy. This will involve conducting wind tunnel tests or CFD simulations to determine how changes in the wake affect the overall drag of the SUV car.

This study researches the streamlined wake of SUV vehicles to distinguish amazing open doors for diminishing drag. Wind tunnel tests and CFD analysis are utilized to examine the stream designs around and behind an assortment of SUV models. The outcomes show that the streamlined wake of SUV vehicles is described by two enormous vortices. These incorporate diminishing the detachment point at the back of the vehicle, streamlining the backside of the vehicle, and utilizing streamlined gadgets like spoilers and diffusers. In this comparative study we designed and developed a prototype of a conventional SUV body with square back to analyse the actual drag and wake behind the vehicle using CFD analysis and Wind Tunnel test. After performing all the tests for conventional SUV, results are documented to compare the results determined with modified body.

KEYWORDS: SUV; CFD; CAD Model; Drag; Base Bleed;

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 10 | October - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

I INTRODUCTION: In the ever-evolving automotive industry, the pursuit of better efficiency, performance, and sustainability is a constant driving force. Among the various factors influencing vehicle design, aerodynamics plays a crucial role in shaping the overall performance of automobiles. Understanding and optimizing the aerodynamic wake of vehicles, particularly SUVs (Sport Utility Vehicles), has become a prominent area of research and development.

As sustainability and environmental concerns become more pressing, the automotive industry strives to develop greener solutions. Enhancing the aerodynamic performance of SUVs can significantly contribute to the reduction of greenhouse gas emissions, promoting an eco-friendlier transportation sector. The implications of aerodynamic improvements are not limited to fuel efficiency and performance gains 5alone. By reducing drag and lift forces, manufacturers can achieve quieter cabins, lower wind noise, and an overall reduction in road noise, contributing to a more comfortable and pleasant driving environment.

Problem Identification: SUVs, with their unique body shapes and versatile applications, have gained immense popularity worldwide. However, their design often presents challenges in terms of aerodynamics due to their larger size, higher ground clearance, and distinctive frontal areas compared to conventional passenger cars. As SUVs continue to dominate the market, engineers and researchers are committed to enhancing their aerodynamic performance to achieve improved fuel efficiency, reduced emissions, and increased stability. This study of aerodynamic wake analysis for SUVs holds great promise in transforming vehicle design and performance.

By understanding the complex flow dynamics surrounding these vehicles, engineers can optimize designs, minimize energy losses, and pave the way for a more sustainable and efficient automotive future. This research aims to shed light on the significant role that aerodynamics plays in shaping the SUVs of tomorrow, as the industry strives for innovation, performance, and environmental responsibility. Moreover, the aerodynamic wake analysis of SUVs also extends to exploring the influence of different operational scenarios, such as varying speeds, crosswinds, and yaw angles. These investigations help engineers optimize the vehicle's shape and implement innovative features to mitigate drag and enhance overall stability, thus enhancing the driving experience and safety.

Problem description: The aerodynamic wake of a vehicle refers to the airflow patterns created as the vehicle moves through the air. These wake patterns are influenced by various factors such as vehicle shape, frontal area, ground clearance, and the presence of add-on features like roof rails, spoilers, and side mirrors. Understanding and optimizing the wake dynamics can significantly impact the vehicle's drag coefficient, lift forces, and overall aerodynamic efficiency.

II DEVELOPMENT OF CAD MODEL

Development of the CAD model plays a major role in this study of aerodynamic wake analysis of SUV, as both for CFD (Computational Fluid Dynamics) analysis and Wind Tunnel test a CAD model is mandatory. In the case of CFD, CAD model is required as a geometry as it was the pre-processing stage in analysis. During the wind tunnel test, a 3D Model is required to analyse the flow over the car body. In this study we planned to develop the 3D model by additive manufacturing whereas a STL file type CAD model is used to generate printer understandable codes. Development of CAD models includes selection of suitable references, design parameters, vehicle specifications and more. In this study, firstly we developed a CAD model for Non-modified SUV (Conventional) cars and modified the design by implementing a bleed vent system where guideways or flow paths are developed and attached to the side walls of the conventional SUV. Throughout his study we used Autodesk Fusion 360 for development of CAD models, where can able to split, combine, and assemble the bodies and components with ease. Stages of development of CAD model have been detailed below.

Development of Conventional SUV CAD model: The process of development of the CAD model requires a selection of suitable workspace. In fusion 360 we can model the car in various workspace like Solid modeling, Surface modeling and freeform modeling. We combined all these workspaces for a specified modification. The outer body, a qualitative design was developed in free form modeling which was also called sculpt modeling to achieve the precise geometry of an SUV. As in other workspaces the tendency to achieve the accuracy will be lesser than free form modeling and it is more complex. The initial sketch of the 2D model has been developed in Paper with the SUV dimensions. We used Mahindra Scorpio to refer to the vehicle dimensions as it was the top selling SUV in India up to September 2023 which was mentioned in the survey released by motor India. The aim is to develop a square back SUV Car model not to develop an exact model of the Mahindra Scorpio. The initial outline sketch of SUV model is shown in figure 1. These Sketches were very helpful while developing the 3D model using CAD software. As it was mentioned earlier, we used Free form modeling workspace to develop the outer body surface of an SUV. We downloaded the exact blueprints of Mahindra Scorpio and Created a required canvas setup at the TOP, SIDE and FRONT by placing the blueprints and scaling to the original dimensions. Then all the three canvases are edited to achieve a proper alignment

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 10 | October - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

of view of the car, whereas the front of the model shows the front blueprint, the side view of the model shows the side of the car in blueprint and same for top. In addition, rear view of the vehicle is required to achieve a proper model. Then decrease the opacity of the canvas to enhance the visibility while controlling nodal points. This canvas setup available in Fusion 360 made us edit controls easily. Using the tools and commands in free form modeling a free form body is created. Repaired all the star, T-spline and edge errors to finish the model. Free form body is converted to the surface body when there are no errors. With the help of surface modeling tools all the surfaces are stitched and reversed normally. Then the surface boundary was converted to solid by using the Boundary fill tool. Then the front and rear wheels are modelled using solid modeling tools like extrude, revolve and cut. Combine all the bodies to form a single car body using combine command, then the surfaces are evaluated to check the free edges or holes and rectify all. Then applied the material to the body and completed some rendering.

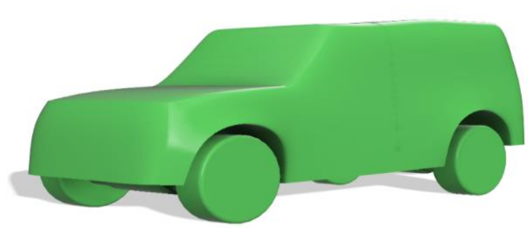

Fig.1 Non-modified SUV CAD model

Development of Modified SUV CAD model: Modified SUV model includes development of guide ways or flow pipes for the base bleed system. The flow pipes were designed to transfer the fluid(air) from the front to the rear of the vehicle to eliminate the wake developed due to turbulence. These guideways are designed in a way that the velocity of incoming air increases due to decrease in diameter of flow paths and separated to three locations beside the vehicle. One facing upwards, which will cover the upper area, another one facing middle and the remaining one facing downwards to cover the whole rear area. This design was inspired by Mercedes Maybach where the air in front of the vehicle was passed to the ground to achieve a better aerodynamic body as it was a lower ground clearance vehicle it won't affect the lift force. In this case the SUV has higher ground clearance, so it should not affect the vehicle by increasing list force. These flow paths are designed in such a way that it won't disturb the other components in vehicles, the inlet opening at the front of the flow path is placed below the headlamps and swept along the inner edges of fender and drag along the cant rail and then passed through the rear fender and the separated three paths are placed below the rear indicators and parking light Flow paths are attached to the walls of outer body. The flow paths at one side of the body is shown in figure 2

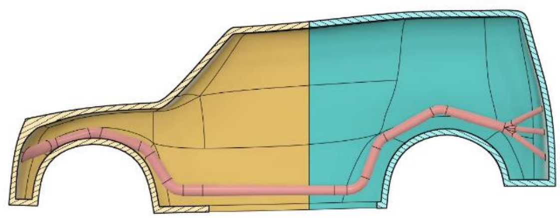

Fig.2 SUV model with flow paths

III CFD ANALYSIS

In aerodynamics, CFD stands for Computational Fluid Dynamics. It is a numerical simulation technique used to analyze and model the behavior of fluids, typically air, as it interacts with objects like aircraft, cars, or buildings. CFD involves solving complex mathematical equations that describe fluid flow, turbulence, and other related phenomena. By simulating fluid flow using CFD, we can gain insights into how air or other fluids move around objects, assess aerodynamic performance, and optimize designs without the need for extensive physical testing. This helps in designing more efficient and aerodynamically sound structures, such as aircraft or vehicles, and is a valuable tool in various fields of engineering and science.

Pre-Processing: Select the particular analysis type based on your venture's targets. This could be static underlying analysis to evaluate the vehicle's primary honesty or transient CFD analysis to concentrate on optimal design. Guarantee that the picked analysis type lines up with your exploration objectives. As our point is to distinguish the streamlined stream, we picked Ansys Familiar (CFD strategy) Start by bringing in the 3D computer aided design model of the downsized SUV vehicle into ANSYS. Guarantee that the computer aided design model is in a viable configuration (e.g., STEP or IGES) and precisely addresses the vehicle's calculation. Make a cross section that discretizes the vehicle's calculation into more modest components. The cross section ought to be already fine to catch complicated subtleties without over-burdening the computational assets. This step is pivotal as the lattice quality straightforwardly influences the exactness of the recreation results. Consider factors like component size and type (tetrahedral, hexahedral, and so forth) based on the analysis type.

Setup and Solution: Put down limit conditions to repeat true situations. Characterize fixed upholds for regions that shouldn't move, for example, the vehicle's wheels contacting the ground. Determine the heaps applied to the vehicle, similar to twist loads for streamlined analysis. Guarantee that limit conditions precisely mirror the imperatives and powers following up on the vehicle. Pick the fitting solver module based on the kind of analysis you mean to perform. For primary analysis, you could decide on "Static Underlying," while streamlined analysis could utilize "Familiar" or other CFD solvers. Design solver settings, including combination measures and time steps on the off chance that your analysis calls for time-subordinate reproductions. Execute the analysis by running the solver. ANSYS will perform calculations based on the characterized math, network, materials, limit conditions, and loads. Be ready to dispense adequate computational assets and time for this step, as intricate reproductions might require a long time to finish.

Finishing these pre-handling steps lays the foundation for the analysis stage. When the analysis is finished, you can continue with post-handling, advancement (if necessary), approval, refinement, last analysis, and documentation and show, as illustrated in the underlying method. Continuously counsel ANSYS documentation and look for master direction while handling complex investigations to guarantee precise and significant outcomes. Post Handling Envision and decipher the outcomes, which might incorporate pressure dissemination, removal, wind current examples, pressure coefficients, and that's only the tip of the iceberg. Generate plots, graphs, and animations to illustrate the findings.

Post-Processing: Record your discoveries in a complete report, including the strategy, input information, reenactment results, and ends. Developing the targets connected with the analysis of the bleed vent, these points by and large aid an extensive evaluation of the vent's primary and useful presentation, considering informed plan upgrades and approval. Assess the reasonableness of materials utilized in the bleed vent development under shifting stacking conditions, recognizing likely upgrades for improved strength and life span. The results obtained through the analysis were evaluated in the results workspace of Ansys Fluent. Geometry preparation and Model setup is shown in figure 3.

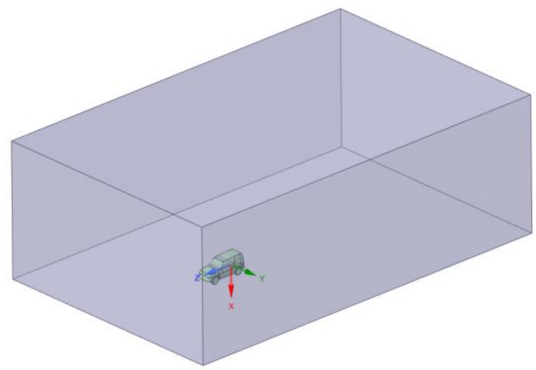

Fig.3 Model with enclosure

IV 3D PRINTING

Fused Deposition Modeling is a generally utilized 3D printing innovation because of its flexibility, cost-viability, and the capacity to make practical models and end-use parts. It's utilized in different ventures, including fabricating, automotive, aviation, medical services and more. The process starts with the creation of a 3D model using computer-aided design (CAD) software. This digital model is sliced into thin, horizontal layers using slicing software, which generates a set of instructions for the 3D printer depending on the type and brand. FDM printers use a wide range of thermoplastic materials, such as PLA, ABS, PETG, and more. The choice of material depends on the specific application and desired properties of the final object. We Used PLA+ to acquire proper bond between layers and material strength.

V WIND TUNNEL TEST

Wind tunnel testing is a crucial process used to determine the aerodynamic characteristics of vehicles like SUVs, which include drag, lift, and side force. Wind tunnel testing is a critical step in the design and engineering of vehicles like SUVs, as it provides valuable insights into their aerodynamic behavior under various conditions, helping manufacturers optimize their designs for performance and safety. The SUV model is typically scaled down to a size suitable for testing in the wind tunnel. The model's design should accurately represent the real vehicle's shape and features, including details such as the mirrors, spoilers, and other aerodynamic elements. The scaled-down model is placed in a wind tunnel, a controlled environment where airflow can be precisely regulated. The wind tunnel has a test section where the model is positioned. Various instruments are attached to the model to collect data, including pressure taps, strain gauges, and velocity measurement devices. Wind tunnel testing allows for precise control over the conditions, including wind speed, air density, and temperature. The model is subjected to different wind speeds and angles of attack to simulate real-world conditions. The wind tunnel used for testing is shown in figure 4

Fig.4 Open circuit Subsonic Suction Wind Tunnel

VI RESULTS

The CFD (Computational Fluid Dynamic) Test using Ansys software helps to find the Drag Coefficient, Total drag, Lift Coefficient, Total lift, Momentum, Force, velocity, Turbulence, etc… Through CFD analysis we analysed the fluid flow characteristics with the help of the Results section in Ansys fluent. Velocity contour, Velocity vector, and Wake developed behind the vehicle were tested and outputs were shown below. A greater understanding of the characteristics of air flow over and around a vehicle, as well as the impact it has on that item, particularly aerodynamic forces, is gained through the utilization of wind tunnel tests by manufacturers and innovators. The Values of drag, lift and Side force for each rise in velocity were tabulated in the figure 5

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 10 | October - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

Frequency (Hz)	Speed (RPM)	Velocity (m/s)	Lift Force (Kg/N)	Drag Force (Kg/N)	Side Force (Kg/N)
6	180	2.5	0.01	0.01	0.01
8	239	4.2	0.01	0.03	0.01
10	298	5.4	0.01	0.04	0.01
12	358	6.7	0.02	0.06	0.02
14	416	8.4	0.02	0.09	0.02
16	476	9.7	0.03	0.12	0.02
18	534	11.3	0.04	0.15	0.02
20	593	12.6	0.04	0.19	0.03
22	625	14.3	0.05	0.24	0.04
24	710	15.6	0.06	0.28	0.05
26	768	17.3	0.07	0.34	0.06
28	827	18.9	0.08	0.39	0.07
30	883	20.2	0.09	0.45	0.08
32	941	21.5	0.11	0.52	0.09
34	998	23.2	0.13	0.58	0.1
36	1055	24.4	0.15	0.64	0.11
38	1112	26.1	0.19	0.73	0.13

Fig.5 Wind tunnel test output

VII CONCLUSION

In conclusion, the imaginative Modified Base bleed system integrated into the Hatchback Vehicle has introduced another period of auto designing greatness. Through a fastidious blend of cutting-edge innovation, computational examination, and functional trial and error, our group has prevailed with regards to enhancing the vehicle's presentation in various viewpoints, eventually rethinking the principles of effectiveness, solace, and maintainability in the auto business.

One of the main accomplishments of our Modified Base bleed system is the astounding decrease in wake, disturbance, and tension at the back part of the vehicle. This accomplishment isn't just a demonstration of the force of state of the art designing yet additionally a crucial forward-moving step in relieving probably the most widely recognized issues that plague customary vehicles. By limiting these aggravations, our system has significantly brought down drag, offering a few important advantages. Most importantly, the diminishing in drag has prompted a significant decrease in fuel utilization. In a period where natural worries and rising fuel costs are at the very front of worldwide conversations, our system offers an unmistakable answer for mitigating both financial and biological weights. The expanded eco-friendliness means less successive visits to the service station and a decreased carbon impression, lining up with the developing interest for maintainable transportation choices.

VIII SUGGESTION

As we plan ahead for transportation, it is developments like the Modified Base bleed system that will make ready for a more supportable and charming driving experience. By decreasing our natural impression, bringing down working expenses, and upgrading the nature of the excursion, we are working on individual lives as well as adding to a cleaner and more productive transportation ecosystem on a worldwide scale.

In the steadily developing universe of car innovation, our Modified Base bleed system remains as a demonstration of what can be accomplished when advancement and designing greatness meet up. It is a demonstration of the force of human creativity in making arrangements that improve our lives while likewise saving our planet for people in the future. With our system, the street ahead isn't simply a way to be voyaged, however an excursion toward a more brilliant and more practical future for all.

IX OUTPUTS

Velocity contour is a visualization tool that displays the distribution of fluid velocity at various points within a computational domain. It's a valuable representation of how fluid flows and moves within a simulated environment. The velocity contour for Non-Modified and Modified model at the centre plane of SUV is shown in figure 6 and 7.

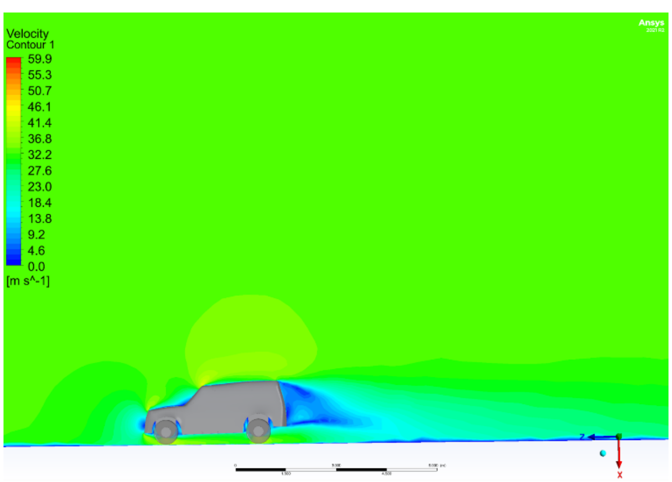

Fig.6 Velocity contour - non-modified

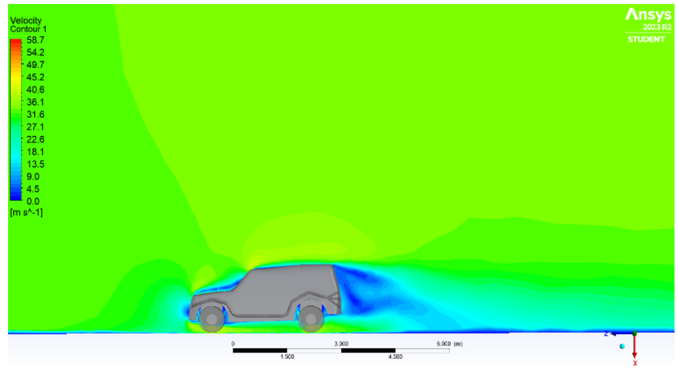

Fig.7 Velocity contour – modified

Velocity vector is a visualization tool that represents the velocity field of a fluid within a computational domain. It displays velocity vectors at different points within the domain to provide a visual representation of the fluid flow patterns. The vector is illustrated at the lateral center plane of SUV with octahedron is shown in following figure 8 and 9.

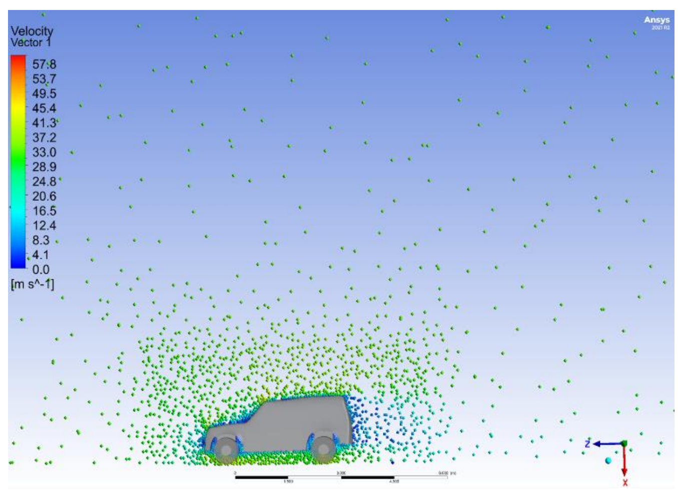

Fig.8 Velocity vector – Non modified

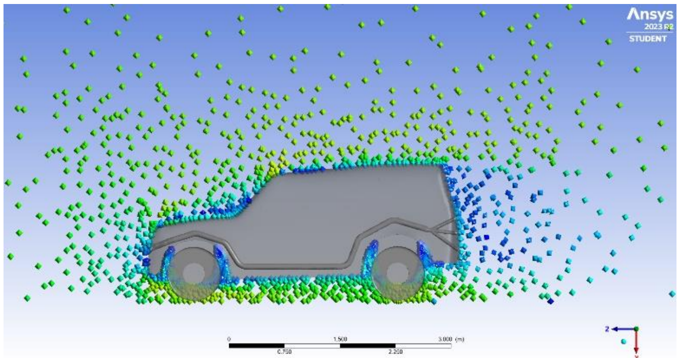

Fig.8 Velocity vector – Modified

The wake developed behind the SUV body is visualized by creating a Plane behind the vehicle at the distance of 10mm from the edge and applied velocity contour in the created plane. The decrease in wake at the bleed areas can be found in the following figure 10 and 11.

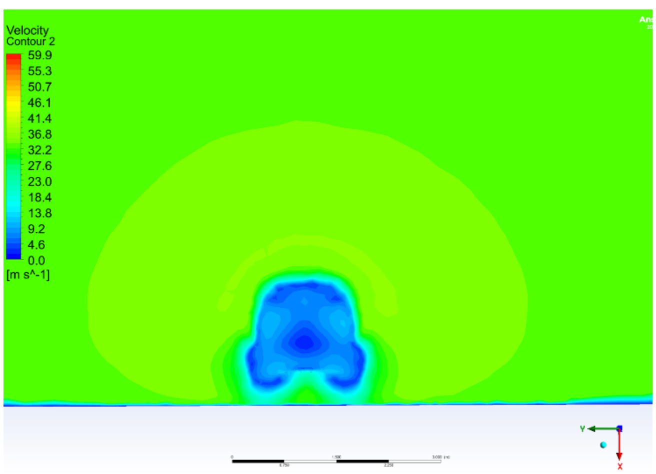

Fig.10 Wake behind 10mm- Non modified

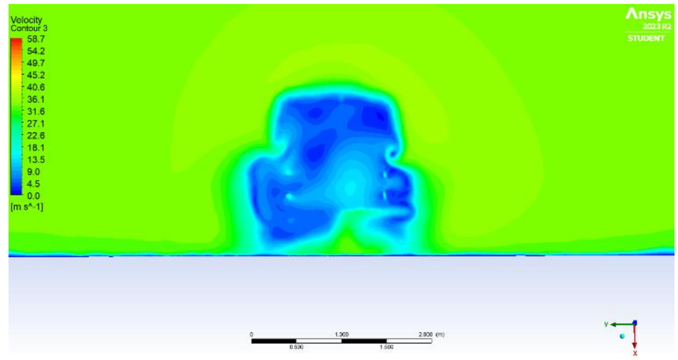

Fig.11 Wake behind 10mm - Modified

REFERANCES

[1]. G. Sivaraj, K. M. Parammasivam and G. Suganya, "Reduction of Aerodynamic Drag Force for Reducing Fuel Consumption in Road Vehicle using Base Bleed," Journal of Applied Fluid Mechanics, Vol. 11, No. 6, pp. 1489-1495,2018.

https://www.researchgate.net/publication/328803849_Available_online_at_ww wjafmonlinenet

[2]. G. Sivaraj, K. M. Parammasivam, M. S. Prasath and D. Lakshmanan, "Numerical Simulation of Hatchback Car with Modified Vehicle Design for the Improvement of Fuel Consumption," Journal of Applied Fluid Mechanics, Vol. 16, No. 9, pp. 1806-1817, 2023.

https://www.jafmonline.net/article_2262.html

[3]. Y.A. Irving Brown, S. Windsor and A.P. Gaylard, "The Effect of Base Bleed and Rear Cavities on the Drag of an SUV," SAE International, 2010-01-0512, 2010.

https://www.researchgate.net/publication/245535456_The_Effect_of_Base_Ble ed_and_Rear_Cavities_on_the_Drag_of_an_SUV

[4]. M. Sathishkumar, Arun Name and Krishn Das Patel, "Experimental Analysis Of Aerodynamic Drag Reduction Of a Hatchback Car by the Rear Spoiler in Wind Tunnel," International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), Vol. 8, 2249-8001,2018.

https://www.academia.edu/37435353/EXPERIMENTAL_ANALYSIS_OF_AE

[5]. Michal Fabian, Robert Hunady, Frantisek Kupec, Tomas Mlaka, "Effect of the Aerodynamic Elements of the Hatchback Tailgate on the Aerodynamic Drag of the Vehicle," Advances in Science and Technology Research Journal 20, 16(6), 73–87,2022.

https://www.researchgate.net/publication/365923164_Effect_of_the_Aerodyna mic_Elements_of_the_Hatchback_Tailgate_on_the_Aerodynamic_Drag_of_the_Vehicle

[6]. Masaki Nakagawa, Frank Michaux, Stephan Kallweit and Kazuhiro Maeda, "Unsteady flow measurements in the wake behind a wind-tunnel car model by using high-speed planar PIV," 11TH INTERNATIONAL SYMPOSIUM ON PARTICLE IMAGE VELOCIMETRY – PIV 15, 2015.

https://www.semanticscholar.org/paper/Unsteady-flow-measurements-in-the- wake-behind-a-car-NakagawaMichaux/b40a7065bff3f918bce79edc2e454b378ea931c4